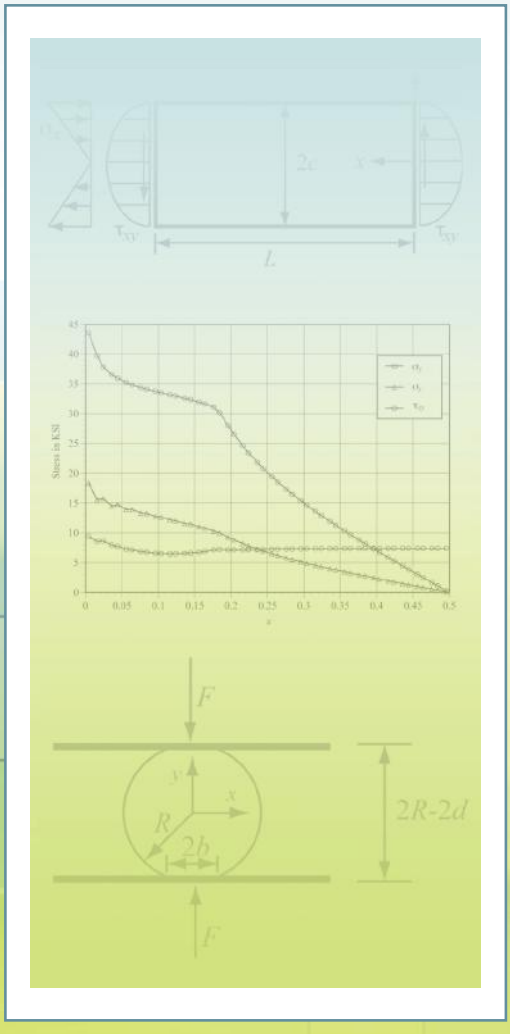
The Finite Element Method for Mechanics of Solids with ANSYS Applications



ELLIS H. DILL

CRC Press
Taylor & Francis Group

The Finite Element Method for Mechanics of Solids with ANSYS Applications

Advances in Engineering

A Series of Reference Books, Monographs, and Textbooks

Series Editor

Haym Benaroya

Department of Mechanical and Aerospace Engineering
Rutgers University

Published Titles:

The Finite Element Method for Mechanics of Solids with ANSYS Applications, *Ellis H. Dill*

Dynamics of Tethered Space Systems, *A. P. Alpatov, V. V. Beletsky, V. I. Dranovskii, V. S. Khoroshilov, A. V. Pirozhenko, H. Troger, and A. E. Zakrzhevskii*

Lunar Settlements, *Haym Benaroya*

Handbook of Space Engineering, Archaeology and Heritage, *Ann Darrin and Beth O'Leary*

Spatial Variation of Seismic Ground Motions: Modeling and Engineering Applications, *Aspasia Zerva*

Fundamentals of Rail Vehicle Dynamics: Guidance and Stability, *A. H. Wickens*

Advances in Nonlinear Dynamics in China: Theory and Applications, *Wenhu Huang*

Virtual Testing of Mechanical Systems: Theories and Techniques, *Ole Ivar Sivertsen*

Nonlinear Random Vibration: Analytical Techniques and Applications, *Cho W. S. To*

Handbook of Vehicle-Road Interaction, *David Cebon*

Nonlinear Dynamics of Compliant Offshore Structures, *Patrick Bar-Avi and Haym Benaroya*

The Finite Element Method for Mechanics of Solids with ANSYS Applications

ELLIS H. DILL

CRC Press
Taylor & Francis Group
Boca Raton London New York

CRC Press is an imprint of the
Taylor & Francis Group, an **informa** business

CRC Press
Taylor & Francis Group
6000 Broken Sound Parkway NW, Suite 300
Boca Raton, FL 33487-2742

Version Date: 20140602

International Standard Book Number-13: 978-1-4398-4584-4 (eBook - PDF)

Contents

Preface

The purpose of this book is to explain application of the finite element method to problems in the mechanics of solids. It is intended for practicing engineers who use the finite element method for stress analysis and for graduate students in engineering who want to understand the finite element method for their research. It is also designed as a textbook for a graduate course in engineering. Application of the finite element method is illustrated by using the ANSYS computer program. Step-by-step instructions for the use of ANSYS Parametric Design Language (APDL) and ANSYS Workbench in more than 40 examples are included.

The required background material in the mechanics of solids is provided so that the work is self-contained for the knowledgeable reader. A more complete treatment of solid mechanics is provided in the book *Continuum Mechanics: Elasticity, Plasticity, Viscoelasticity* by Ellis H. Dill (CRC Press, 2007). References to that book are noted in this book on an applicable page by a footnote (Dill: specific referral detail).

This book is not intended as a detailed reference book on the use of the ANSYS system. However, Chapters 15 and 16 contain detailed steps for the application of ANSYS in numerous examples, which will enable the user to become fairly proficient in the use of this software. The new user should begin with one of the tutorials provided by ANSYS or with one of the elementary books listed in the bibliography in Chapters 15 and 16. This book was written using Version 12.1. However, the examples in Chapters 15 and 16 can be executed using either Version 12 or Version 13. I do not pretend to present a detailed analysis of finite element as implemented by ANSYS. I do not have access to their computer coding. I believe that the elements they are using are essentially the same as those presented here, although they may differ in some details.

I have attempted to cover only the essentials of the subject and to provide the tools necessary for comprehension of the technical literature and the commercial finite element programs. I apologize in advance to all of the originators of this material who are not referenced. I have long ago forgotten where I learned the theory.

BoCheng Jin helped with the preparation of the manuscript and provided many corrections to it. Of course, any remaining errors are mine alone.

Author

Ellis H. Dill obtained his BS, MS, and PhD from the University of California (Berkeley) in civil engineering. He taught aeronautical engineering at the University of Washington (Seattle) from 1956 to 1977. He was dean of engineering at Rutgers, the State University of New Jersey, from 1977 to 1998. Dr. Dill is currently a university professor at Rutgers, teaching mechanical and aerospace engineering. His principal research areas include aircraft structures, analysis of plates and shells, solid mechanics, and the finite element method of stress analysis. He can be reached by email at dill@rutgers.edu.

1 Finite Element Concepts

1.1 INTRODUCTION

The finite element method (FEM) has developed along two paths. From the mathematical point of view, it is a method of constructing a function that makes the potential energy a minimum. From the engineering point of view, it is a method of assembling structural elements, which can be separately analyzed, into a global equation of equilibrium for the structure. The mathematical point of view makes the FEM a special form of the Rayleigh–Ritz method, which has a long history. The modern FEM may be said to have begun with Courant in 1943.[1] His paper had little impact because the method was not practical until the development of digital computers in the 1950s. This approach has now been extensively explored by mathematicians and placed on a sound mathematical basis. Precise studies of error analysis and convergence proofs are available.[2–5] However, the study of the mathematical foundations, involving Sobolev spaces, is beyond the scope of this book.

The emphasis in this book is on the direct stiffness method in which the unknowns are the displacements of particular points, and to a lesser degree on the mixed (U-P) method, in which the mean stress is a primary variable. However, Chapter 3 contains the fundamental variational theorems underlying the general mixed and hybrid methods that seemed to show great promise but have not achieved prominence in practical engineering analysis. The most significant omission is the new meshless method of analysis that has been recently developed.[6]

The analysis of structures by dividing them into elements, such as beams, stringers, shear panels, and so forth, which can be separately analyzed, has been developed over the past hundred years into a standard method of engineering analysis. Organization of the calculations using matrix algebra was widely developed, from about 1950 onward, as computers became available that made such computational methods practical.[7] A landmark paper on the application of the direct stiffness formulation to continuum problems was published by Turner, Clough, Martin, and Topp in 1956.[8] The method was later named "finite element" method by Clough,[9] in contrast to the finite difference method that was widely used for solution of continuum problems at that time.

From the viewpoint of the structural engineer, the analysis of a structure is accomplished by writing equations for the assembly of structural elements that describe

(1) Compatibility or continuity of the deformations
(2) Equilibrium of the contact forces at joints
(3) Force–deformation relations for the elements

In the direct stiffness method, from which the FEM evolved, continuity of the displacements (and rotations) is achieved by expressing all of the elements and joint

displacements in a single global coordinate system and then equating the displacements where elements are joined. The equilibrium of forces acting on the joints is then easily expressed by using the same global coordinate system for the contact forces from the joined structural elements. The force–deformation relation is a relation expressing the forces acting on an element as a linear function of the joint displacements. The coefficient matrix is called the element stiffness matrix for the element. Elimination of the element forces from the equilibrium equations leads to a single linear algebraic equation for the external forces in terms of the joint displacements. The coefficient matrix is called the global stiffness matrix.

In this book, the emphasis will be on FEM as a systematic method for constructing a function that makes the potential energy a minimum. However, the concepts that have arisen from matrix formulations of structural analysis will also be used. For example, the direct addition, or merge, of element stiffness matrices will be an important concept.

I will first summarize the direct stiffness method of structural analysis in more detail from the viewpoint of the structural engineer.

1.2 DIRECT STIFFNESS METHOD

A structure can be modeled as an assembly of *elements* that are joined at discrete points called *nodes*. For example, a truss consists of axial force elements joined at their ends. A frame consists of beam elements. An airplane consists of frames, stringers, spars, and shear panels. A mechanical component can be modeled as an assembly of solid elements joined at the corners.

We can introduce a global rectangular Cartesian coordinate system for components of displacement of the joints and the external forces applied to the joint. The term "displacements" includes rotations, which are considered to be "generalized displacements." All elements connected to a common joint share the displacements of that joint. Let us denote the components of joint displacement in the global Cartesian coordinate system by D_1, D_2, D_3, etc., and the corresponding components of external force by F_1, F_2, F_3, etc. The forces may be either reactions or given external loads. The subscripts can be assigned in any order, but each component is given a distinct label and the indices range consecutively from 1 to N, with N being the total number of components of joint displacements for the structure. We call each displacement component D_i a *degree of freedom* (DOF).

For each element, we must establish a relation between the internal forces exerted by the joints on the element and the displacements of the joints to which the element is attached. This is accomplished by a stress analysis of the element that is done before we began to analyze the articulated structure. For example, for a truss member in the elastic range, the axial force is proportional to the elongation. For a beam element in the elastic range, the joint forces consist of forces and moments that are linearly related to the displacements and rotations of the ends of the beam. The moments are regarded as "generalized forces." For an element m, which behaves elastically and has only small displacements, the relation between joint displacements and element forces (components in the global Cartesian system) is expressed by a linear equation:

$$f_i^m = \sum_j k_{ij}^m \ D_j, \quad i, j \in I_m. \tag{1.1}$$

This relation involves only a subset of joint displacements: i is in the index set I_m of displacements for the member m. The summation implied by the repeated index j is over the set I_m. The forces f_i^m are the components of force exerted by the joint on the element, acting in the same direction as the corresponding joint displacement D_i, and *they are numbered by the same index number i*. The matrix $\mathbf{k}^m$ with elements k_{ij}^m is therefore a square matrix called the stiffness matrix of element m. It is always a symmetric matrix:

$$k_{ij}^m = k_{ji}^m. \tag{1.2}$$

The element stiffness relation 1.1 can be written as a matrix equation:

$$\mathbf{f}^m = \mathbf{k}^m \mathbf{D}^m, \text{ no sum on } m. \tag{1.3}$$

However, the indices (ij) denote the related displacement component and do not follow the standard row–column matrix notation. The element stiffness matrix is a symmetric square matrix with the number of columns and rows equal to the number of displacement components of the joints attached to member m.

1.2.1 Merging the Element Stiffness Matrices

We must now set forth the requirement that the forces applied to the joint by the elements are in equilibrium with the external forces applied to the joint. Resolving the external forces into the same components as we used for the joint displacements and member forces, we have, for joint n, the relation

$$F_i = \sum_{m \in \mathcal{M}_i} f_i^m, \ i \in \mathcal{D}_n. \tag{1.4}$$

The range of m is over the index set $\mathcal{M}_i$ of members that share the DOF D_i. The range of i is over the index set $\mathcal{D}_n$ of displacement components for the joint n. Substituting the element stiffness relations 1.1 into the joint equilibrium Equations 1.4, we obtain the global relation between external force and joint displacement. This operation is to be done for all joints to obtain one equation for each DOF, $i = 1$ to N:

$$\begin{aligned} F_i &= \sum_{m \in \mathcal{M}_i} \left(\sum_{j \in I_m} k_{ij}^m D_j \right) \\ &= \sum_{k \in C_i} \left(\sum_{m \in \mathcal{M}_i} k_{ik}^m \right) D_k. \end{aligned} \tag{1.5}$$

The summation on k in the second term is over the set C_i of those DOFs that are connected to the ith DOF by some member, that is, the *connectivity* of the structure. By definition, no two joints share the same F_i or D_i, and the total number of such force and displacement components is N, that is, $i = 1$ to N.

The summation on k in the last term can be extended to the full range of displacements,

$$F_i = \sum_{k=1}^{N} K_{ik} D_k, \tag{1.6}$$

by defining $K_{ik} = 0$ for those k such that D_i and D_k are not connected by any member:

$$K_{ik} = \begin{cases} \sum_{m \in \mathcal{M}_i} k_{ik}^m \text{ for } k \in C_i, \\ 0 \text{ for } k \notin C_i. \end{cases} \tag{1.7}$$

Then, in matrix notation,

$$\mathbf{F} = \mathbf{KD} \tag{1.8}$$

This summation of the element stiffness matrices is called *merging* of the matrices to form the global stiffness matrix. In the global $N \times N$ stiffness matrix $\mathbf{K}$ with terms K_{ik}, the index i becomes the row number and the index k becomes the column number of the term.

In Equation 1.7, we are merely adding together all of the terms with common indices from each of the element matrices. We can start by setting all of the terms in the global stiffness matrix $\mathbf{K}$ to zero. We then take any one element and *add* all of the terms from the element stiffness matrix directly into the global stiffness matrix at the appropriate location. Then we go to the next element and repeat the addition of terms from the element stiffness matrix into the global stiffness matrix. This is the process that gave rise to the terminology "merging the stiffness matrices." It is an efficient numerical method for forming the global stiffness matrix. Henceforth, when we indicate a summation of element stiffness matrices, the summation will be understood to mean that element stiffness matrices are *merged*.

The external forces $\mathbf{F}$ consist of the externally applied loads and reactions and the inertial forces. If we approximate the inertial forces by lumping the mass at the joints, the inertial force is $(-m_i \ddot{D}_i)$, no sum on i, for each DOF. The general form of Equation 1.8 including inertial forces is

$$\mathbf{F} - \mathbf{M}\ddot{\mathbf{D}} = \mathbf{KD} \tag{1.9}$$

where $\mathbf{M}$ is a diagonal matrix with the lumped masses m_i on the diagonal.

1.2.2 Augmenting the Element Stiffness Matrix

It is sometimes helpful to visualize geometrically the process of forming the global stiffness matrix **K** from the element stiffness matrices. One can imagine that each element stiffness matrix is increased in size to match the global stiffness by inserting zero terms for all terms, other than those terms corresponding to the indices i and j occurring in the element array k_{ij}^m, to obtain an $N \times N$ element matrix $\hat{k}_{ij}^m$. Equation 1.7 then expresses the ordinary matrix addition:

$$\mathbf{K} = \sum_m \hat{\mathbf{k}}^m, \tag{1.10}$$

where the summation is over the totality of elements and $\hat{\mathbf{k}}^m$ is the element matrix augmented by zeros. This has several advantages conceptually. One may think of each element stiffness matrix as written on a sheet of paper with the terms k_{ij}^m entered into the row and column of the global array as dictated by indices i and j. The sheets of paper are laid on top of one another and the elements are added that lie in the same position. However, this is not a good plan for computations because it involves manipulating a lot of zeros.

1.2.3 Stiffness Matrix Is Banded

The geometrical concept of merging the element stiffness matrices is also helpful in understanding the banded nature of the global stiffness matrix. If we are forming the row of **K** corresponding to say F_1, then the only elements that will contribute terms to this row are those attached to the joint having the DOF D_1, and the only terms that those elements can contribute to **K** will be those for the columns corresponding to the displacements of the other DOFs associated with those elements. Consequently, only those terms in that row that are contributed by the elements sharing the DOF D_1 can be nonzero. Beyond a certain column number, all of the remaining terms of the row F_1 are zero. Thus, the nonzero terms are confined to a band emanating from the diagonal elements of **K**. By numbering the joint displacements judiciously, we can minimize the width of this band and confine the nonzero terms to a relatively small band around the diagonal of **K**.

1.3 THE ENERGY METHOD

The calculations can also be described in terms of the potential energy. The strain energy of each element is

$$\mathcal{U}^m = \sum_{i \in I_m} \sum_{j \in I_m} k_{ij}^m D_i D_j = \frac{1}{2} (\mathbf{D}^m)^{\mathrm{T}} \mathbf{k}^m \mathbf{D}^m, \tag{1.11}$$

where $\mathbf{D}^m$ denotes the column matrix of the DOFs for the element m. We will see later how this formula for the strain energy is derived from the field equations of

linear elasticity. The strain energy of the collection of elements is the sum of the stain energy of each one:

$$\begin{aligned}\mathcal{U} &= \sum_m \mathcal{U}^m \\ &= \sum_m \left(\sum_{i \in I_m} \sum_{j \in I_m} k_{ij}^m D_i D_j \right) \\ &= \sum_{i=1}^{N} \sum_{j=1}^{N} K_{ij} D_i D_j\end{aligned} \tag{1.12}$$

where **K** is the merge of the element matrices. In matrix notation,

$$\mathcal{U} = \frac{1}{2} \mathbf{D}^{\mathrm{T}} \mathbf{K} \mathbf{D} \tag{1.13}$$

where the global stiffness matrix **K** is the result of merging the element stiffness matrices $\mathbf{k}^m$ as described above.

In the case of given loads **F** applied to the joints, the potential of the external loads is the negative of the force times the displacement. The potential energy for the system is therefore

$$\mathcal{P} = \mathcal{U} - \mathbf{F}^{\mathrm{T}} \mathbf{D} = \frac{1}{2} \mathbf{D}^{\mathrm{T}} \mathbf{K} \mathbf{D} - \mathbf{F}^{\mathrm{T}} \mathbf{D}. \tag{1.14}$$

The condition for a minimum of the potential energy is

$$\frac{\partial \mathcal{P}}{\partial D_i} = 0 \tag{1.15}$$

where *i* ranges over all of the (unknown) DOFs. Applying this condition to the total potential 1.14 gives the global Equations 1.8 of equilibrium:

$$\mathbf{K} \mathbf{D} = \mathbf{F}. \tag{1.16}$$

The energy method is one that we will exploit for the general formulation.* It offers several advantages. First, the calculations are automatic once the element stiffness matrices have been determined. Second, approximate solutions are readily formulated by simply deriving an approximate potential energy. Third, the mathematical studies of error and convergence often make explicit use of the minimum of the potential energy in equilibrium problems.

* Actually, we will use the first derivative which is called the virtual work formula.

The energy method can be extended to explicitly include the inertial forces by introducing the kinetic energy. If the mass is lumped at a node point (joint), the kinetic energy associated with D_j is

$$\mathcal{T}_j = \frac{1}{2} m_j \dot{D}_j \dot{D}_j, \quad \text{(no sum)}. \tag{1.17}$$

The total kinetic energy of the system is the sum over the number of DOFs:

$$\mathcal{T} = \sum_j \mathcal{T}_j. \tag{1.18}$$

Combining 1.17 and 1.18, we find

$$\mathcal{T} = \frac{1}{2} \dot{\mathbf{D}}^{\mathrm{T}} \mathbf{M} \dot{\mathbf{D}}, \tag{1.19}$$

where the global mass matrix is the result just a diagonal matrix consisting of the individual lumped masses. The equations of motion can be derived from Lagrange's equations

$$\frac{\mathrm{d}}{\mathrm{d}t} \frac{\partial \mathcal{T}}{\partial \dot{D}_j} - \frac{\partial \mathcal{T}}{\partial D_j} = -\frac{\partial \mathcal{P}}{\partial D_j}. \tag{1.20}$$

this leads to the general equations of motion for the discrete system:

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{F} \tag{1.21}$$

This is a set of linear ordinary differential equations that can be solved by standard methods. However, because of the banded nature of **M** and **K**, special techniques can be used to reduce the computational effort.

1.4 TRUSS EXAMPLE

A truss is a collection of axial force members that are joined at the ends in such a way that there is no restraint on the relative rotation of the members at the joint. Each member can be considered an *element*. The joints are the *nodes*. The components of displacement of the joints are the *degrees of freedom* of the structure.

Let us consider the truss shown in Figure 1.1, which has three elements and four nodes numbered as shown. The DOFs are labeled as shown in Figure 1.2. In this case, the total number of DOFs is $N = 8$. The forces exerted on the elements by the nodes are shown in Figure 1.3, with positive directions as shown.

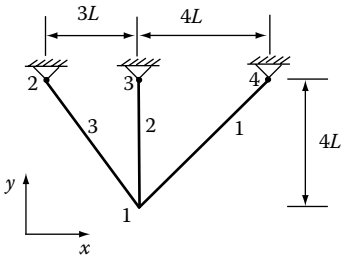


FIGURE 1.1 Truss example.

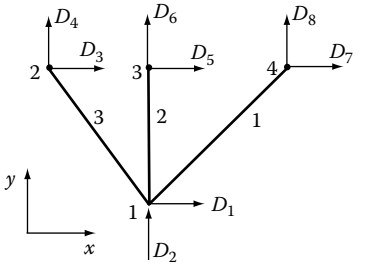


FIGURE 1.2 Degrees of freedom for truss.

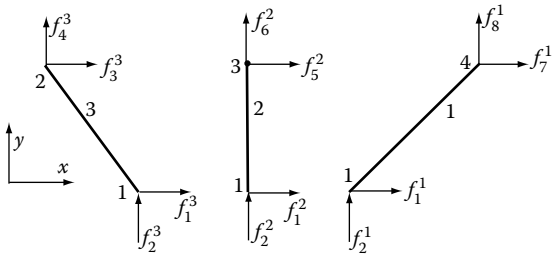


FIGURE 1.3 Element forces.

The force–displacement relations for member 1 are

$$\begin{aligned} f_i^1 &= k_{i1}^1 D_1 + k_{i2}^1 D_2 + k_{i7}^1 D_7 + k_{i8}^1 D_8, \\ i &\in I_1, \quad I_1 = (1,\ 2,\ 7,\ 8), \end{aligned} \tag{1.22}$$

or

$$f_i^1 = \sum_{j \in I_1} k_{ij}^1 D_j. \tag{1.23}$$

For member 2:

$$f_i^2 = \sum_{j \in I_2} k_{ij}^2 D_j, \quad i \in I_2, \quad I_2 = (1, 2, 5, 6). \tag{1.24}$$

For member 3:

$$f_i^3 = \sum_{j \in I_3} k_{ij}^3 D_j, \quad i \in I_3, \quad I_3 = (1, 2, 3, 4). \tag{1.25}$$

The forces exerted by the external world and by the elements on the nodes are shown in Figure 1.4. The sign convention is that f_i^m is in the positive coordinate direction on the element and therefore in the negative coordinate direction on the joint. The equilibrium of forces for node 1 requires that

$$F_i = f_i^1 + f_i^2 + f_i^3 = \sum_{m \in \mathcal{M}_1} f_i^m, \quad i \in \mathcal{D}_1, \quad \mathcal{D}_1 = (1, 2), \ \mathcal{M}_1 = (1, 2, 3). \tag{1.26}$$

The equilibrium of forces for node 2 requires that

$$F_i = \sum_{m \in \mathcal{M}_2} f_i^m, \quad i \in \mathcal{D}_2, \quad \mathcal{M}_2 = (3), \quad \mathcal{D}_2 = (3, 4). \tag{1.27}$$

The equilibrium of forces for node 3 requires that

$$F_i = \sum_{m \in \mathcal{M}_3} f_i^m, \quad i \in \mathcal{D}_3, \quad \mathcal{M}_3 = (2), \quad \mathcal{D}_3 = (5, 6). \tag{1.28}$$

The equilibrium of forces for node 4 requires that

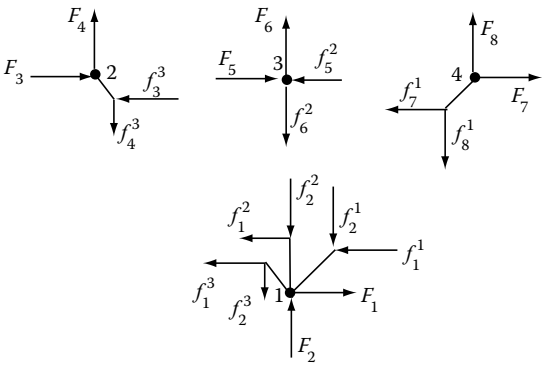


FIGURE 1.4 Forces on nodes.

$$F_i = \sum_{m \in \mathcal{M}_4} f_i^m, \quad i \in \mathcal{D}_4, \quad \mathcal{M}_4 = (1), \quad \mathcal{D}_4 = (7, 8). \tag{1.29}$$

Substitution of the force–displacement relations for the elements provides the force–displacement relations for the assembled structure. For example,

$$\begin{aligned} F_1 &= k_{11}^1 D_1 + k_{12}^1 D_2 + k_{17}^1 D_7 + k_{18}^1 D_8 \\ &+ k_{11}^2 D_1 + k_{12}^2 D_2 + k_{15}^2 D_5 + k_{16}^2 D_6 \\ &+ k_{11}^3 D_1 + k_{12}^3 D_2 + k_{13}^3 D_3 + k_{14}^3 D_4. \end{aligned} \tag{1.30}$$

Adding the common factors for each DOF (merging the stiffness matrices) gives eight force–displacement relations. For the first DOF,

$$F_1 = K_{11} D_1 + K_{12} D_2 + K_{13} D_3 + K_{14} D_4 + K_{15} D_5 + K_{16} D_6 + K_{17} D_7 + K_{18} D_8, \tag{1.31}$$

where

$$\begin{aligned} K_{11} &= k_{11}^1 + k_{11}^2 + k_{11}^3 \\ K_{12} &= k_{12}^1 + k_{12}^2 + k_{12}^3 \\ K_{13} &= k_{13}^3 \\ K_{14} &= k_{14}^3 \\ K_{15} &= k_{15}^2 \\ K_{16} &= k_{16}^2 \\ K_{17} &= k_{17}^1 \\ K_{18} &= k_{18}^1. \end{aligned} \tag{1.32}$$

For the third DOF, occurring at node 2,

$$F_3 = f_3^3 = k_{31}^3 D_1 + k_{32}^3 D_2 + K_{33}^3 D_3 + k_{34}^3 D_4. \tag{1.33}$$

The connectivity sets for the DOFs are

$$\begin{aligned} &C_1 = (1, 2, 3, 4, 5, 6, 7, 8), C_2 = (1, 2, 3, 4, 5, 6, 7, 8), C_3 = (1, 2, 3, 4), C_4 = (1, 2, 3, 4), \\ &C_5 = (1, 2, 5, 6), C_6 = (1, 2, 5, 6), C_7 = (1, 2, 7, 8), C_8 = (1, 2, 7, 8). \end{aligned} \tag{1.34}$$

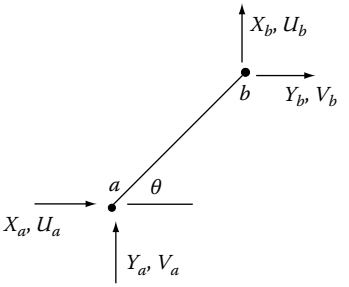


FIGURE 1.5 Truss element.

In particular, there is no element connecting DOFs 3 and 4 to DOFs 5, 6, 7, and 8. Therefore, $K_{35} = K_{36} = K_{37} = K_{38} = 0$, and so forth.

We still have to analyze the truss element in order to determine the element stiffness matrices. Each member may be inclined to the x-axis by an angle θ as shown in Figure 1.5. For end a, let X_a and Y_a be the components of the member force f_a. Let U_a and V_a denote the components of axial displacement D_a. A similar notation is used at end b.

Let $\alpha = \cos\theta$ and $\beta = \sin\theta$. Then, $X_a = \alpha f_a$ and $Y_a = \beta f_a$. The axial displacement of end a is $D_a = \alpha U_a + \beta V_a$. A similar notation is used for end b. Using Equations 1.44 and 1.45, we find the stiffness relation for the element*:

$$\begin{bmatrix} X_a \\ Y_a \\ X_b \\ Y_b \end{bmatrix} = \frac{AE}{l} \begin{bmatrix} \alpha\alpha & \alpha\beta & -\alpha\alpha & -\alpha\beta \\ \alpha\beta & \beta\beta & -\alpha\beta & -\beta\beta \\ -\alpha\alpha & -\alpha\beta & \alpha\alpha & \alpha\beta \\ -\alpha\beta & -\beta\beta & \alpha\beta & \beta\beta \end{bmatrix} \begin{bmatrix} U_a \\ V_a \\ U_b \\ V_b \end{bmatrix}. \tag{1.35}$$

For the truss shown in Figure 1.1, Equation 1.35 applies to each member with the appropriate values of α and β. Remember that angle θ is measured counterclockwise from the x-axis in each case. For the case when A and E are the same for all elements, the element stiffness matrices are as follows:

$$\mathbf{k}^1 = \frac{AE}{8L\sqrt{2}} \begin{array}{c} \begin{matrix} D_1 & D_2 & D_7 & D_8 \end{matrix} \\ \begin{bmatrix} +1 & +1 & -1 & -1 \\ +1 & +1 & -1 & -1 \\ -1 & -1 & +1 & +1 \\ -1 & -1 & +1 & +1 \end{bmatrix} \end{array}. \tag{1.36}$$

* ANSYS element LINK1.

$$\mathbf{k}^2 = \frac{AE}{4L}\begin{array}{c} \begin{matrix} D_1 & D_2 & D_5 & D_6 \end{matrix} \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & +1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & +1 \end{bmatrix} \end{array}. \tag{1.37}$$

$$\mathbf{k}^3 = \frac{AE}{125L}\begin{array}{c} \begin{matrix} D_1 & D_2 & D_3 & D_4 \end{matrix} \\ \begin{bmatrix} +9 & -12 & -9 & +12 \\ -12 & +16 & +12 & -16 \\ -9 & +12 & +9 & -12 \\ +12 & -16 & -12 & +16 \end{bmatrix} \end{array}. \tag{1.38}$$

Merging the three element stiffness matrices gives the global stiffness matrix:

$$\mathbf{K} = \frac{AE}{L}\begin{array}{c} \begin{matrix} D_1 & D_2 & D_3 & D_4 & D_5 & D_6 & D_7 & D_8 \end{matrix} \\ \begin{bmatrix} \frac{9}{125}+\frac{1}{8\sqrt{2}} & \frac{-12}{125}+\frac{1}{8\sqrt{2}} & \frac{-9}{125} & \frac{12}{125} & 0 & 0 & -\frac{1}{8\sqrt{2}} & -\frac{1}{8\sqrt{2}} \\ \frac{-12}{125}+\frac{1}{8\sqrt{2}} & \frac{16}{125}+\frac{1}{4}+\frac{1}{8\sqrt{2}} & \frac{12}{125} & \frac{-16}{125} & 0 & -\frac{1}{4} & -\frac{1}{8\sqrt{2}} & -\frac{1}{8\sqrt{2}} \\ \frac{-9}{125} & \frac{12}{125} & \frac{9}{125} & \frac{-12}{125} & 0 & 0 & 0 & 0 \\ \frac{12}{125} & \frac{-16}{125} & \frac{-12}{125} & \frac{16}{125} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -\frac{1}{4} & 0 & 0 & 0 & \frac{1}{4} & 0 & 0 \\ -\frac{1}{8\sqrt{2}} & -\frac{1}{8\sqrt{2}} & 0 & 0 & 0 & 0 & \frac{1}{8\sqrt{2}} & \frac{1}{8\sqrt{2}} \\ -\frac{1}{8\sqrt{2}} & -\frac{1}{8\sqrt{2}} & 0 & 0 & 0 & 0 & \frac{1}{8\sqrt{2}} & \frac{1}{8\sqrt{2}} \end{bmatrix} \end{array}. \tag{1.39}$$

The global stiffness matrix has zero determinant so Equation 1.8 does not have a unique solution for given loads **F**. This is to be expected because the unsupported structure allows rigid translation and rotation and sometimes collapse as a mechanism. If the supported structure can act as a mechanism, it is said to be kinematically unstable.

Using the condition of zero displacement at nodes 2, 3, and 4, we find the equilibrium equations for the supported structure:

$$\frac{AE}{L}\begin{bmatrix} \frac{1}{8\sqrt{2}}+\frac{9}{125} & \frac{1}{8\sqrt{2}}-\frac{12}{125} \\ \frac{1}{8\sqrt{2}}-\frac{12}{125} & \frac{1}{8\sqrt{2}}+\frac{1}{4}+\frac{16}{125} \end{bmatrix}\begin{bmatrix} U_1 \\ V_1 \end{bmatrix}=\begin{bmatrix} X_1 \\ Y_1 \end{bmatrix}. \tag{1.40}$$

The coefficient matrix is called the reduced stiffness matrix.

The joint displacement can now be calculated for given external forces X_1 and Y_1 by solving Equation 1.40. For $Y_1 = 0$,

$$U_1 = 6.2397\frac{X_1 L}{AE}, \quad V_1 = .101835\frac{X_1 L}{AE}. \tag{1.41}$$

This example may be used as a test problem (Section 15.3).

1.5 AXIALLY LOADED ROD EXAMPLE

Next, let us consider another simple example that illustrates the ideas of the direct stiffness formulation. A straight rod is loaded by a force applied to one end and supported at the other end (Figure 1.6). The solution of the static problem is trivial, but the matrix formulation may still be useful if additional loads are applied at various points along the rod, or if the material properties vary along the rod. The problem is not so trivial if the loads vary with time and inertial forces are included.

We will formulate approximate equations governing the motion of the rod by using a finite element model. To this end, we divide the rod into of a number of elements. Suppose, for example, we use two elements that are joined at the midpoint of the rod (Figure 1.7). One-half of the mass of each element is lumped at the joint at the end of the element as a first approximation for the inertial forces. For a more accurate solution, one simply increases the number of elements, tending to the exact solution as the number of elements tends to infinity.

The axial (x direction) displacement at each point is denoted by D_i, $i = 1, 2, 3$. The contact forces on element n are denoted by f_i^n, where i takes on the values of the joint numbers bounding the element (Figure 1.8). The element properties are approximated as uniform over each element. This may require a larger number of elements for a satisfactory analysis if the properties are varying rapidly.

We can analyze the stress state for each massless element. The equations of linear elasticity for the element are

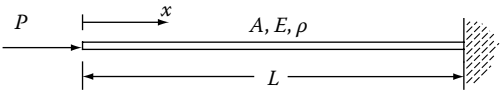


FIGURE 1.6 Axial force applied to a rod.

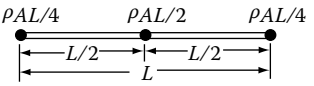


FIGURE 1.7 A finite element model of the rod.

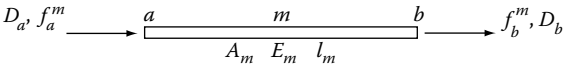


FIGURE 1.8 Typical rod element.

$$\varepsilon = \frac{\partial u}{\partial x},$$

$$\frac{\partial \sigma}{\partial x} = 0, \tag{1.42}$$

$$\sigma = E\varepsilon,$$

where E is the modulus of elasticity of the material. The solution is

$$u = \frac{x_b - x}{l} D_a + \frac{x - x_a}{l} D_b. \tag{1.43}$$

The axial force at any point can then be calculated: $f_a = -\sigma A$ and $f_b = +\sigma A$, where A is the area of the rod. In matrix form, for a generic element,

$$\begin{bmatrix} f_a^m \\ f_b^m \end{bmatrix} = \begin{bmatrix} k_{aa}^m & k_{ab}^m \\ k_{ba}^m & k_{bb}^m \end{bmatrix} \begin{bmatrix} D_a \\ D_b \end{bmatrix}, \tag{1.44}$$

where a and b range from 1 to 2 for $m = 1$, whereas a and b range from 2 to 3 for $m = 2$. The specific values for the elements of the stiffness matrix are

$$\begin{aligned} k_{aa}^m = k_{bb}^m &= \frac{AE}{l}, \\ k_{ab}^m = k_{ba}^m &= -\frac{AE}{l}. \end{aligned} \tag{1.45}$$

Thus, the force deformation relation for element 1 is

$$\begin{bmatrix} f_1^1 \\ f_2^1 \end{bmatrix} = \begin{bmatrix} k_{11}^1 & k_{12}^1 \\ k_{21}^1 & k_{22}^1 \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \end{bmatrix}, \tag{1.46}$$

and for element 2,

$$\begin{bmatrix} f_2^2 \\ f_3^2 \end{bmatrix} = \begin{bmatrix} k_{22}^2 & k_{23}^2 \\ k_{32}^2 & k_{33}^2 \end{bmatrix} \begin{bmatrix} D_2 \\ D_3 \end{bmatrix}, \tag{1.47}$$

with the terms k_{ab}^m defined as stated above. Notice that no element index is required for the displacement components because they are the displacements of the joint to which the element is attached. And the indices on k_{ab}^m match the displacement and force components, not the location in the element stiffness matrix.

The equilibrium of forces for the joint j is expressed by

$$F_j - m_j \ddot{D}_j = \sum_m f_j^m, \text{ (no sum on } j), \tag{1.48}$$

where F_j is the external load acting on the joint, m_j is the mass that has been lumped at the joint, a superposed dot indicates the time derivative, and the summation extends over the elements that are connected to the joint j. Substitution of the force–deformation relations 1.46 and 1.47 into the equilibrium relation 1.48 with $l = L/2$ leads to the global equation of equilibrium, relating the external loads to the joint displacements:

$$\begin{bmatrix} \frac{\rho AL}{4} & 0 & 0 \\ 0 & \frac{\rho AL}{2} & 0 \\ 0 & 0 & \frac{\rho AL}{4} \end{bmatrix} \begin{bmatrix} \ddot{D}_1 \\ \ddot{D}_2 \\ \ddot{D}_3 \end{bmatrix} + \begin{bmatrix} \frac{2AE}{L} & -\frac{2AE}{L} & 0 \\ -\frac{2AE}{L} & \frac{4AE}{L} & -\frac{2AE}{L} \\ 0 & -\frac{2AE}{L} & \frac{2AE}{L} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \\ D_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix}. \tag{1.49}$$

Alternatively, using 1.33, 1.14, and 1.20, we obtain once more the governing Equations 1.49 by the energy method. In matrix notation,

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{F}. \tag{1.50}$$

At each joint, either F_j or Dj is given and the equations have to be solved for the other quantity. In the particular case shown in Figure 1.6, $D_3 = 0$. There are therefore three equations in three unknowns (D_1, D_2, F_3), where F_3 is the reaction at the support. It is convenient to divide the relations into two sets. First, solve for the unknown displacements corresponding to the given forces:

$$\begin{bmatrix} \frac{\rho AL}{4} & 0 \\ 0 & \frac{\rho AL}{2} \end{bmatrix} \begin{bmatrix} \ddot{D}_1 \\ \ddot{D}_2 \end{bmatrix} + \begin{bmatrix} \frac{2AE}{L} & -\frac{2AE}{L} \\ \frac{-2AE}{L} & \frac{4AE}{L} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \end{bmatrix} = \begin{bmatrix} P \\ 0 \end{bmatrix}. \tag{1.51}$$

Then, solve for the reactions at the joints with zero displacement:

$$\begin{bmatrix} 0 & 0 \end{bmatrix} \begin{bmatrix} \ddot{D}_1 \\ \ddot{D}_2 \end{bmatrix} + \begin{bmatrix} 0 & -\dfrac{2AE}{L} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \end{bmatrix} = [F_3]. \tag{1.52}$$

The zero terms are shown for completeness. If the inertial loads are neglected ($\ddot{D}_1 = 0$, $\ddot{D}_2 = 0$), then the solution to Equation 1.51 is

$$\begin{bmatrix} D_1 \\ D_2 \end{bmatrix} = \frac{PL}{2AE} \begin{bmatrix} 2 \\ 1 \end{bmatrix}. \tag{1.53}$$

We will consider methods of solution of the equations later. At this time, we simply note that the problem is reduced to a set of linear ordinary differential equations in time, or to a set of linear algebraic equations if the inertial forces are neglected.

1.5.1 Augmented Matrices for the Rod

To illustrate the augmented element matrices, let us label the rows and columns of the global stiffness matrix with the corresponding force and displacement components. The augmented stiffness matrix for element 1 is

$$\hat{\mathbf{k}}^1 = \begin{array}{c} \\ F_1 \\ F_2 \\ F_3 \end{array} \begin{array}{c} \begin{array}{ccc} D_1 & D_2 & D_3 \end{array} \\ \begin{bmatrix} \dfrac{2AE}{L} & -\dfrac{2AE}{L} & 0 \\ -\dfrac{2AE}{L} & \dfrac{2AE}{L} & 0 \\ 0 & 0 & 0 \end{bmatrix} \end{array}. \tag{1.54}$$

The augmented stiffness matrix for element 2 is

$$\hat{\mathbf{k}}^2 = \begin{array}{c} \\ F_1 \\ F_2 \\ F_3 \end{array} \begin{array}{c} \begin{array}{ccc} D_1 & D_2 & D_3 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 \\ 0 & \dfrac{2AE}{L} & -\dfrac{2AE}{L} \\ 0 & -\dfrac{2AE}{L} & \dfrac{2AE}{L} \end{bmatrix} \end{array}. \tag{1.55}$$

The global stiffness matrix **K** is the sum of these two augmented matrices in the usual sense of matrix addition.

$$\mathbf{K} = \sum_m \hat{\mathbf{k}}^m . \tag{1.56}$$

However, I emphasize again that this is not a practical method for real engineering problems because of the large number of zeros that must be handled.

1.5.2 Merge of Element Matrices for the Rod

We will now illustrate the process of forming the global stiffness matrix by merging the element matrices:

$$\mathbf{K} = \sum_m \mathbf{k}^m . \tag{1.57}$$

First, construct the global stiffness matrix with all terms equal to zero:

$$\mathbf{K} = \begin{matrix} \\ F_1 \\ F_2 \\ F_3 \end{matrix} \begin{matrix} \begin{matrix} D_1 & D_2 & D_3 \end{matrix} \\ \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \end{matrix} . \tag{1.58}$$

After merging the stiffness matrix of element 1, the global stiffness matrix is

$$\mathbf{K} = \begin{matrix} \\ F_1 \\ F_2 \\ F_3 \end{matrix} \begin{matrix} \begin{matrix} D_1 & D_2 & D_3 \end{matrix} \\ \begin{bmatrix} \frac{2AE}{L} & -\frac{2AE}{L} & 0 \\ -\frac{2AE}{L} & \frac{2AE}{L} & 0 \\ 0 & 0 & 0 \end{bmatrix} \end{matrix} . \tag{1.59}$$

After merging the stiffness matrix of element 2 to this matrix, the final global stiffness matrix is

$$\mathbf{K} = \begin{matrix} \\ F_1 \\ F_2 \\ F_3 \end{matrix} \begin{matrix} \begin{matrix} D_1 & D_2 & D_3 \end{matrix} \\ \begin{bmatrix} \frac{2AE}{L} & -\frac{2AE}{L} & 0 \\ -\frac{2AE}{L} & \frac{4AE}{L} & -\frac{2AE}{L} \\ 0 & -\frac{2AE}{L} & \frac{2AE}{L} \end{bmatrix} \end{matrix} . \tag{1.60}$$

The symbols D_i and F_i are written along side of the columns and rows in order to identify the related component of force and displacement. They are only labels to assist in the merge process. A computer program will always assume that the third row corresponds to F_3 for example, without any labels. If we are forming the row of **K** corresponding to say F_1, then the only elements that will contribute terms to this row are those attached to joint 1, and the only terms that they can contribute will be those for the columns corresponding to the displacements of the other end of the element. In this simple example, only element number one can contribute to the row for F_1, and it is connected only to joints 1 and 2, so the only entries will be to the columns for D_1 and D_2, regardless of the number of elements used to model the rod. There is no contribution to the term (F_1, D_3), or any column beyond D_3, which are therefore zero, even if there were more elements. Starting with the diagonal element and counting to the right, there will be only two nonzero elements in each row. This is the called the half bandwidth of **K**.

1.6 FORCE METHOD

In the preceding formulation, the joint displacements are the fundamental unknowns. Before stored-program digital computers became available, it was more common to think of the element forces as fundamental unknowns. For any assemblage of rods, the resultant axial for each rod was regarded as the fundamental unknown. The difference in view point can be made clear by an example. Let us consider, for example, the truss shown Figure 1.9. All members have area A and modulus E.

Let D_1 and D_2 denote the displacements in the direction of the applied forces F_1 and F_2. Let ε_i denote the increment in length of element i. Let S_i denote the tensile force for element i. The equilibrium relations for the loaded joint are

$$\begin{bmatrix} -\frac{1}{\sqrt{2}} & 0 & \frac{3}{5} \\ -\frac{1}{\sqrt{2}} & -1 & -\frac{4}{5} \end{bmatrix} \begin{bmatrix} S_1 \\ S_2 \\ S_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \text{ or } \mathbf{BS} = \mathbf{F}. \tag{1.61}$$

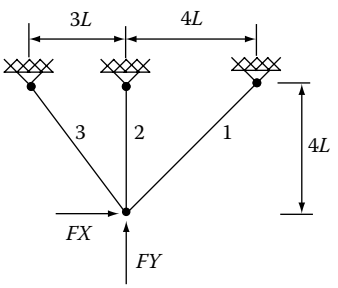


FIGURE 1.9 Statically indeterminate truss.

The geometric relations between extensions of the elements and the joint displacements are

$$\begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{bmatrix} = \begin{bmatrix} -\frac{1}{\sqrt{2}} & -\frac{1}{\sqrt{2}} \\ 0 & -1 \\ \frac{3}{5} & -\frac{4}{5} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \end{bmatrix} \text{ or } \boldsymbol{\varepsilon} = \mathbf{AD}. \tag{1.62}$$

The constitutive relations between the extensions of the elements and the axial forces are

$$\begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{bmatrix} = \begin{bmatrix} \frac{4\sqrt{2L}}{AE} & 0 & 0 \\ 0 & \frac{4L}{AE} & 0 \\ 0 & 0 & \frac{5L}{AE} \end{bmatrix} \begin{bmatrix} S_1 \\ S_2 \\ S_3 \end{bmatrix} \text{ or } \boldsymbol{\varepsilon} = \mathbf{fS} \tag{1.63}$$

where **f** is the *flexibility matrix* of the structure.

The theorem of virtual work that we will develop later states that the internal work equals the external work, $\mathbf{S}^T\boldsymbol{\varepsilon} = \mathbf{F}^T\mathbf{D}$, for every equilibrium system (**S**, **F**) and every compatible system ($\boldsymbol{\varepsilon}$, **D**). That is, $\mathbf{S}^T\mathbf{AD} = \mathbf{S}^T\mathbf{B}^T\mathbf{D}$ for every **S** and **D**. Therefore, $\mathbf{A} = \mathbf{B}^T$, as the explicit example shows.

In general, there will be s stress parameters and d displacement parameters. The matrix **B** is $d \times s$. If $s = d$, the equilibrium equations can be solved for **S** unless the det **B** = 0. If the determinant is zero, the system is kinematically unstable. If $d > s$, then **S** is overdetermined. This implies the existence of displacement fields with zero generalized strains, and the structure is unstable. If $d < s$, as in the example, the equilibrium Equations 1.61 have more unknowns S_i than equations, and the structure is said to be *statically indeterminate*. The general solution to 1.61 is then of the form

$$\mathbf{S} = \mathbf{b}_0\mathbf{F} + \mathbf{b}_1\mathbf{R}. \tag{1.64}$$

The elements of **R** are $s - d = r$ in number and are called *redundant forces*. In this example, there is one redundant force R_1. We can choose the force in member 3 as the redundant (or any other member): $S_3 = R_1$. Then, we find from 1.61 that

$$\mathbf{b}_0 = \begin{bmatrix} -\sqrt{2} & 0 \\ 1 & -1 \\ 0 & 0 \end{bmatrix}, \quad \mathbf{b}_1 = \begin{bmatrix} 3\sqrt{2}/5 \\ -7/5 \\ 1 \end{bmatrix}. \tag{1.65}$$

The coefficient matrices are such that

$$\mathbf{B}\mathbf{b}_0 = 1 \tag{1.66}$$

and

$$\mathbf{B}\mathbf{b}_1 = 0. \tag{1.67}$$

Since $\mathbf{A} = \mathbf{B}^T$, we have from 1.62 and 1.67 the compatibility relation

$$\mathbf{b}_1^T\boldsymbol{\varepsilon} = 0. \tag{1.68}$$

Substituting 1.63 into 1.68,

$$\mathbf{c}_0\mathbf{F} + \mathbf{c}_1\mathbf{R} = 0 \tag{1.69}$$

where

$$\mathbf{c}_0 = \mathbf{b}_1^T\mathbf{f}\mathbf{b}_0, \quad \mathbf{c}_1 = \mathbf{b}_1^T\mathbf{f}\mathbf{b}_1. \tag{1.70}$$

Therefore,

$$\mathbf{R} = -\mathbf{c}_1^{-1}\mathbf{c}_0\mathbf{F}. \tag{1.71}$$

Substituting 1.71 into 1.64 yields the formula for solution by the redundant force method:

$$\mathbf{S} = \mathbf{b}\mathbf{F}, \tag{1.72}$$

where

$$\mathbf{b} = \mathbf{b}_0 - \mathbf{b}_1\mathbf{c}_1^{-1}\mathbf{c}_0. \tag{1.73}$$

There exists standard algorithms for constructing the solution 1.64 of the equilibrium equations, and the redundant force method is a viable procedure for stress analysis provided someone has developed a computer code to automate the calculations.

It may be noticed that the stiffness method, which was introduced first, avoided altogether the question of redundant forces. The two methods are, however, equivalent for the same structural model. The stiffness formulation can be recovered by eliminating S and ε from Equations 1.61 to 1.63:

$$\mathbf{K}\mathbf{D} = \mathbf{F} \tag{1.74}$$

where

$$\mathbf{K} = \mathbf{B}\mathbf{f}^{-1}\mathbf{B}^{\mathrm{T}}. \tag{1.75}$$

This **K** is the reduced stiffness matrix that one obtains after applying the displacement boundary conditions.

1.7 OTHER STRUCTURAL COMPONENTS

1.7.1 Space Truss

In general, elements of the truss may have any spatial orientation with respect to the global coordinate system. The axial displacement at each end is then resolved into components along the three coordinate axes, so there are 3 DOFs at each end.

If the numbering scheme follows that of Figure 1.10, the DOFs associated with node i will be $\mathbf{D}^{\mathrm{T}} = [D_{3i-2}D_{3i-1}D_{3i}]$ as shown in Figure 1.10. The corresponding element and nodal forces are numbered in the same manner: $\mathbf{F}^{\mathrm{T}} = [F_{3i-2}F_{3i-1}F_{3i}]$. For element m that connects nodes 1 and 2, we have the force–displacement relation:

$$\begin{bmatrix} f_1^m \\ f_2^m \\ f_3^m \\ f_4^m \\ f_5^m \\ f_6^m \end{bmatrix} = \frac{A_m E_m}{L_m} \begin{bmatrix} \alpha^2 & \alpha\beta & \alpha\gamma & -\alpha^2 & -\alpha\beta & -\alpha\gamma \\ \alpha\beta & \beta^2 & \beta\gamma & -\alpha\beta & -\beta^2 & -\beta\gamma \\ \alpha\gamma & \beta\gamma & \gamma^2 & -\alpha\gamma & -\beta\gamma & -\gamma^2 \\ -\alpha^2 & -\alpha\beta & -\alpha\gamma & \alpha^2 & \alpha\beta & \alpha\gamma \\ -\alpha\beta & -\beta^2 & -\beta\gamma & \alpha\beta & \beta^2 & \beta\gamma \\ -\alpha\gamma & -\beta\gamma & -\gamma^2 & \alpha\gamma & \beta\gamma & \gamma^2 \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \\ D_3 \\ D_4 \\ D_5 \\ D_6 \end{bmatrix}, \tag{1.76}$$

where α, β, and γ are the direction cosines of the rod. An assembly of such elements is a three-dimensional truss. Each joint is a node with 3 DOFs.

1.7.2 Beams and Frames

In general, a rod element may be subject to bending and twist as well as axial deformations and may be called a rod, beam, or beam–column element. We will suppose

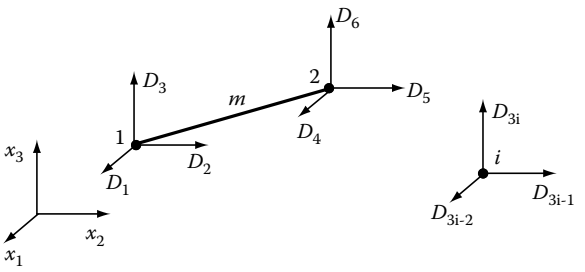


FIGURE 1.10 Space truss element.

here that all loads are applied at the ends of the beam element. There are 6 DOFs at each end: 3 components of displacement and 3 components of rotation. The first three being components of displacement along the coordinate axes, and the second three being rotations about the axes. By elementary mechanics of materials, we can establish the relation between the generalized forces on the ends of the rod and the generalized displacements of the ends of the rod. An assembly of such elements is a three-dimensional space frame. Each joint is a node with 6 DOFs.

We will only formulate the equations governing the lateral displacements of a beam limited to bending in a plane. For bending without extension in the plane, the only displacement component of significance is the lateral displacement $v(x, t)$. The sign convention for beam theory is shown in Figure 1.11.

For small displacements, each beam element is governed by the following equations for the rotation α, the curvature κ, the shear force V, and the bending moment M:

$$\begin{aligned} &\alpha = \frac{\partial v}{\partial x}, \quad \kappa = \frac{\partial \alpha}{\partial x}, \\ &\frac{\partial V}{\partial x} + p = \rho \frac{\partial^2 v}{\partial t^2}, \quad \frac{\partial M}{\partial x} + V = 0, \\ &M = EI\kappa. \end{aligned} \tag{1.77}$$

We will consider the case when the loads are applied only at the ends of the element (node points) so that $p = 0$, and the mass is concentrated at the node points ($\rho = 0$).

The nodal displacements and rotations of the ends of the beam (Figure 1.12) are

$$v_1 = v(0), \ \theta_1 = \alpha(0), \ v_2 = v(L), \ \theta_2 = \alpha(L). \tag{1.78}$$

The general solution of 1.77 is

$$\begin{aligned} V &= C_1, \\ M &= -C_1 x + C_2, \\ EI\alpha &= -C_1 \frac{x^2}{2} + C_2 x + C_3, \\ EIv &= -C_1 \frac{x^3}{6} + C_2 \frac{x^2}{2} + C_3 x + C_4. \end{aligned} \tag{1.79}$$

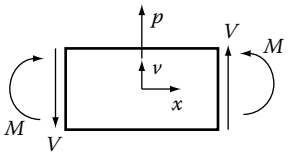


FIGURE 1.11 Beam sign convention.

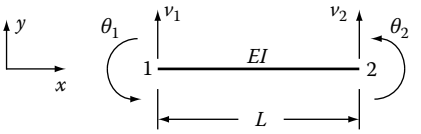


FIGURE 1.12 Definition of nodal DOF.

Therefore,

$$
\begin{aligned}
C_4 &= EIv_1, \\
C_3 &= EI\theta_1, \\
C_2 &= -\frac{6EI}{L^2}v_1 - \frac{4EI}{L}\theta_1 + \frac{6EI}{L^2}v_2 - \frac{2EI}{L}\theta_2 \\
C_1 &= -\frac{12EI}{L^3}v_1 - \frac{6EI}{L^2}\theta_1 + \frac{12EI}{L^3}v_2 - \frac{6EI}{L^2}\theta_2.
\end{aligned}
\tag{1.80}
$$

The ends of the beam element are nodes in the finite element analysis. The sign convention for nodal forces is shown in Figure 1.13. The forces and moments on the ends of the rod are

$$
f_1 = -V(0),\ m_1 = -M(0),\ f_2 = V(L),\ m_2 = M(L). \tag{1.81}
$$

Therefore,

$$
\begin{bmatrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{bmatrix} =
\begin{bmatrix}
\frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\
\frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\
-\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\
\frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L}
\end{bmatrix}
\begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix}. \tag{1.82}
$$

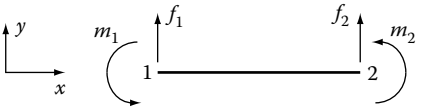


FIGURE 1.13 Generalized forces on a beam element.

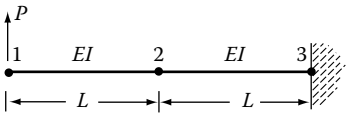


FIGURE 1.14 Cantilever beam problem.

This establishes the element stiffness matrix. Note that this is the exact solution of the beam equations for the static equilibrium problem neglecting shear deformations.

As an example, consider the cantilever beam supported and loaded as shown in Figure 1.14.

We will divide the whole beam into two elements of equal length and assume the same cross section and material for both. After merging the two stiffness matrices we have

$$\begin{bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \\ F_3 \\ M_3 \end{bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 \\ -12 & -6L & 24 & 0 & -12 & 6L \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 \\ 0 & 0 & -12 & -6L & 12 & -6L \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix}. \tag{1.83}$$

The given load and support conditions are

$$v_3 = \theta_3 = 0, \quad F_1 = P, \quad M_1 = F_2 = M_2 = 0. \tag{1.84}$$

The solution of 1.83 for the conditions 1.84 is

$$\begin{aligned} v_1 &= \frac{8}{3}\frac{PL^3}{EI}, \quad \theta_1 = -2\frac{PL^2}{EI}, \\ v_2 &= \frac{5}{6}\frac{PL^3}{EI}, \quad \theta_2 = -\frac{3}{2}\frac{PL^2}{EI}, \\ F_3 &= -P, \quad M_3 = 2PL. \end{aligned} \tag{1.85}$$

1.7.2.1 General Beam Equations

By merging the stiffness matrix (1.44) for axial deformations with the stiffness matrix (1.82) for bending, we obtain the stiffness matrix for beam–columns*:

* ANSYS element BEAM3.

$$\begin{array}{c} \\ f_{x1} \\ f_{y1} \\ m_1 \\ f_{x2} \\ f_{y2} \\ m_2 \end{array} \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & \theta_1 & u_2 & v_2 & \theta_2 \end{array} \\ \begin{bmatrix} k_a & 0 & 0 & -k_a & 0 & 0 \\ 0 & 12k_b & 6Lk_b & 0 & -12k_b & 6Lk_b \\ 0 & 6LK_b & 4Lk_b & 0 & -6Lk_b & 2Lk_b \\ -k_a & 0 & 0 & k_a & 0 & 0 \\ 0 & -12k_b & -6Lk_b & 0 & 12k_b & -6Lk_b \\ 0 & 6Lk_b & 2Lk_b & 0 & -6Lk_b & 4Lk_b \end{bmatrix} \end{array} \tag{1.86}$$

where

$$k_a = \frac{AE}{L}, \quad k_b = \frac{EI}{L^3}. \tag{1.87}$$

The stiffness matrix (1.86) is expressed in a local coordinate system with the x-axis along the beam and lying in the x–y plane. A coordinate transformation can be applied for application to any global coordinate system.

The rod may also be subject to twisting by a torque at the ends. The corresponding stiffness matrix can be merged with Equation 1.86 to obtain the stiffness matrix for a general rod element:

$$\begin{array}{c} \\ f_{x1} \\ f_{y1} \\ m_{z1} \\ m_{x1} \\ f_{x2} \\ f_{y2} \\ m_{z2} \\ m_{x2} \end{array} \begin{array}{c} \begin{array}{cccccccc} u_1 & v_1 & \theta_{z1} & \theta_{x1} & u_2 & v_2 & \theta_{z2} & \theta_{x2} \end{array} \\ \begin{bmatrix} k_a & 0 & 0 & 0 & -k_a & 0 & 0 & 0 \\ 0 & 12k_b & 6Lk_b & 0 & 0 & -12k_b & 6Lk_b & 0 \\ 0 & 6Lk_b & 4Lk_b & 0 & 0 & -6Lk_b & 2Lk_b & 0 \\ 0 & 0 & 0 & k_c & 0 & 0 & 0 & -k_c \\ -k_a & 0 & 0 & 0 & k_a & 0 & 0 & 0 \\ 0 & -12k_b & -6Lk_b & 0 & 0 & 12k_b & -6Lk_b & 0 \\ 0 & 6Lk_b & 2Lk_b & 0 & 0 & -6Lk_b & 4Lk_b & 0 \\ 0 & 0 & 0 & -k_c & 0 & 0 & 0 & k_c \end{bmatrix} \end{array} \tag{1.88}$$

where

$$k_c = \frac{GJ}{L} \tag{1.89}$$

is the torsional stiffness. We have used an appropriate change in notation for 1.86 and 1.88 that is often needed to clearly state the components of force and displacement. The fully general case would involve bending about the y-axis as well.* That matrix,

* ANSYS element BEAM4.

similar to 1.82, can be merged with 1.88. We have implicitly assumed that the x–y axes are the principal axes of the cross section.

1.7.3 Plates and Shells

Structural elements that have one dimension that is very small compared to the other two, such as a sheet of paper or an eggshell, are called a plate if the element is flat and a shell if the element is curved. We call these two-dimensional elements. For elements with four edges, the corners are usually chosen as nodes. At each node there are 6 DOFs, as for the general beam element. In order to determine the stiffness matrix for such elements, we need the theory of plates and shells. We will consider the theory of bending of plates later. Shells are not covered in this book.

1.7.4 Two- or Three-Dimensional Solids

Machine parts generally have all dimensions of comparable magnitude. They must be treated as three-dimensional bodies. A stress analysis involves the solution of a problem in the theory of elasticity, plasticity, or viscoelasticity. A typical finite element is brick shaped. Each corner is generally a node and the DOFs at each node are the three components of displacement at the node. In the subcase of plane stress or plane strain, the analysis involves only a two-dimensional region. The elements are then typically taken to be triangles or quadrilaterals with nodes at the corners, and 2 DOFs at each node. The derivation of the finite element equations, the stiffness matrix for the element, merging of the element stiffness matrices, and the solution of the finite element equations is the main subject of the remainder of this book.

1.8 PROBLEMS

1. Divide the axially loaded rod shown in Figure 1.15 into two elements of equal length with node numbers as shown in Figure 1.16.
   (a) Write the stiffness matrix for each element.
   (b) Merge the element stiffness matrices to obtain the global 3 by 3 stiffness matrix for the unconstrained rod.

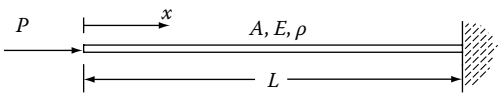


FIGURE 1.15 Axial force rod.

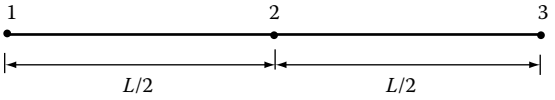


FIGURE 1.16 Finite element model.

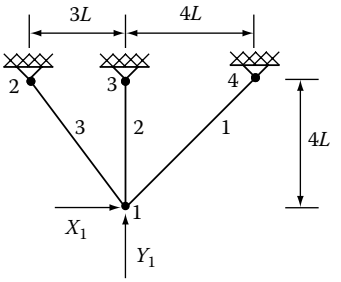


FIGURE 1.17 Truss configuration.

2. Divide the axially loaded rod shown in Figure 1.15 into three elements of equal length.
   (a) Write the stiffness matrix for each element showing relevant displacement and force numbers.
   (b) Merge the element stiffness matrices to obtain the global 4 by 4 stiffness matrix for the unconstrained rod.
3. Divide the axially loaded rod (Figure 1.15) into three elements of equal length.
   (a) Form the global stiffness matrix, the global mass matrix, and the global force matrix.
   (b) Apply the boundary conditions and determine the reduced set of equations to be solved for displacements, but do not solve the resulting set of equations.
4. The truss shown in Figure 1.17 has $A = 1$, $L = 1$, $E = 30 \times 10^6$, $X_1 = 100$, $Y_1 = 0$. Solve for the nodal displacements using ANSYS and compare with the exact solution. (See Section 15.3.)
5. Consider the bending of a cantilever beam modeled with two elements (Figure 1.18).
   (a) Determine the global stiffness matrix $\mathbf{K}$ by merging the element matrices.
   (b) Apply the support conditions to determine the reduced stiffness matrix $\mathbf{K}_0$.
6. The cantilever beam shown Figure 1.18 has a square cross section, $h = 2$, $w = 1$, $A = 2$, $I = 2/3$, $E = 1000$, $L = 5$, $P = 1$. Solve the following beam bending problem using ANSYS (see Section 15.4).
   (a) Print list of UY at each node.
   (b) Submit list of moments at the right end of each element from an Element Data Table.

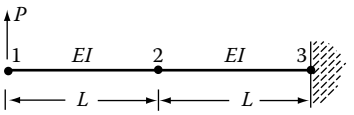


FIGURE 1.18 Bending of a beam.

(c) Submit list of bending stress at the right end of each element from an Element Data Table.
(d) Compare with numerical solution with the exact solution.

REFERENCES

1. Courant, R., Variational methods for the solution of problems of equilibrium and vibration, *Bull. Am. Math. Soc*., 49, 1–43, 1943.
2. Strang, G., and G. J. Fix, *An Analysis of the Finite Element Method*, Prentice-Hall, Inc., Englewood Cliffs, NJ, 1973.
3. Oden, J. T., and J. N. Reddy, *An introduction to the Mathematical Theory of Finite Elements*, John Wiley, New York, 1976.
4. Oden, J. T., and G. F. Carey, *Finite Elements: Mathematical Aspects*, Prentice-Hall, Englewood Cliffs, NJ, 1983.
5. Brenner, S. C., and L. R. Scott, *The Mathematical Theory of Finite Element Methods*, Springer, New York, 3rd ed. 2008.
6. Atluri, S. A., *The Meshless Method (MLPG) for Domain & BIE Discretizations*, Tech Science Press, Forsyth, GA, 2004.
7. Argyris, J., and S. Kelsey, *Energy Theorems and Structural Analysis*, Butterworth Scientific Publications, London, 1960.
8. Turner, M. J., R. Clough, H. Martin, and L. Topp, *Stiffness and Deflection Analysis of Complex Structures*, *J. Aero. Sci.*, 23, no. 9, Sept., 1956, pp. 805–823.
9. Clough, R., The finite element in plane stress, *Proc. 2nd ASCE Conf. on Electronic Computation,* Pittsburgh, PA, Sept. 1960.

BIBLIOGRAPHY

Cook, R. D., *Concepts and Applications of Finite Element Analysis*, John Wiley, New York, 1974.

Gallagher, R. H., *Finite Element Analysis*, Prentice-Hall, Englewood Cliffs, NJ, 1975.

Martin, H. C., *Introduction to Matrix Methods of Structural Analysis*, McGraw-Hill, New York, 1966.

Przemieniecki, J. S., *Theory of Matrix Structural Analysis*, McGraw-Hill, New York, 1968.

Zienkiewicz, O. C., *The Finite Element Method in Structural and Continuum Mechanics*, McGraw-Hill, London, 1967.

2 Linear Elasticity

2.1 BASIC EQUATIONS

In this chapter we will summarize the set of differential equations that govern the motion of elastic bodies experiencing small deformations. This set of differential equations, together with the boundary and initial conditions, is a well-posed mathematical problem that we wish to solve via numerical methods. It is assumed that the reader is familiar with the derivation and use of these relations. They are presented here in rectangular Cartesian components without derivation. Index notation with summation convention is used. English letter indices have the range (1, 2, 3) and Greek letter indices have the range (1, 2).

2.1.1 Geometry of Deformation

We consider a material body that occupies a region $\mathcal{V}$ of space with a boundary $\mathcal{S}$. A particle of material has coordinates $x_k = (x,y,z)$ (before deformation) and experiences a displacement vector with components $u_k = (u,v,w)$ that are functions of the coordinates x_k and time t. Extensions and shears of material fibers are determined by the strain tensor, which has components ε_{km} that are related to the components of the displacement vector by

$$\varepsilon_{km} = \frac{1}{2}\left(\frac{\partial u_k}{\partial x_m} + \frac{\partial u_m}{\partial x_k} \right) \equiv u_{(k,m)}. \tag{2.1}$$

We will also use the alternate notation:

$$\begin{aligned} \varepsilon_{11} &= \varepsilon_x, \ 2\varepsilon_{12} = \gamma_{xy}, \\ \varepsilon_{22} &= \varepsilon_y, \ 2\varepsilon_{23} = \gamma_{yz}, \\ \varepsilon_{33} &= \varepsilon_z, \ 2\varepsilon_{31} = \gamma_{zx}. \end{aligned} \tag{2.2}$$

The extension (normal strain) of a material fiber that is initially along the x_1 axis is ε_{11}. The shear (shear strain) of a pair of fibers that are initially along the x_1 and x_2 axis is $\gamma_{xy} = 2\varepsilon_{12}$. That is, the physical shear is twice the tensor component. Each component u_i of the displacement vector is a continuous function of the coordinates x_i and the function is continuously differentiable except possibly at isolated points, lines, or surfaces. Any continuous displacement field together with the strain tensor determined by Equation 2.1 is called a *compatible system*.

2.1.2 Balance of Momentum

The state of stress is characterized by the stress tensor with components τ_{km}. The components τ_{km} are functions of the coordinates x_k and time t. At each time t the functions are continuous and differentiable in $\mathcal{V}$ except possibly at isolated points, lines, or surfaces. The balance of linear momentum is expressed by

$$\frac{\partial \tau_{ij}}{\partial x_j} + b_i = \rho \ddot{u}_i. \tag{2.3}$$

The balance of angular momentum is expressed by

$$\tau_{ij} = \tau_{ji}. \tag{2.4}$$

We will also use the alternate notation:

$$\begin{aligned} \tau_{11} &= \tau_{xx} = \sigma_x, \ \tau_{12} = \tau_{xy}, \\ \tau_{22} &= \tau_{yy} = \sigma_y, \ \tau_{23} = \tau_{yz}, \\ \tau_{33} &= \tau_{zz} = \sigma_z, \ \tau_{31} = \tau_{zx}. \end{aligned} \tag{2.5}$$

Now, consider a closed surface bounding all or part of the material. Let n_i denote the components of the unit vector normal to the surface and directed toward the exterior of the body. The components of the force per unit area T_i exerted on the body at the bounding surface by the surrounding material, or by the exterior world, is related to the stress tensor at each point on the surface by

$$T_i = \tau_{ij} n_j. \tag{2.6}$$

The vector with components T_i is called the stress vector or traction vector. The stress vector is continuous across each interior surface. A smooth stress field τ_{ij} together with the surface tractions T_i determined by 2.6 is called an *equilibrium system*.

2.1.3 Virtual Work

An alternative statement of the balance of momentum is provided the theorem, or principle, of *virtual work*. This relation can be derived as follows.

Let $\overline{u}_i$ and $\overline{\varepsilon}_{ij}$ denote any compatible system. Multiplying 2.3 by $\overline{u}_i$, summing on i, and integrating by parts gives

$$\int_S T_i \overline{u}_i \, \mathrm{d}A + \int_{\mathcal{V}} (b_i - \rho \ddot{u}_i) \overline{u}_i \, \mathrm{d}V = \int_{\mathcal{V}} \tau_{ij} \overline{\varepsilon}_{ij} \, \mathrm{d}V. \tag{2.7}$$

We have used $\tau_{ij} = \tau_{ji}$ and $T_i = \tau_{ij} n_j$. Equation 2.7 has a physical interpretation: The first term is the work of surface tractions T_i moving through displacements $\overline{u}_i$ and the second is the work of the total body forces. The right side expresses the work of the internal stresses τ_{ij}, which are in equilibrium with the surface tractions T_i and the total body force. That is, the external work equals internal work for a body in equilibrium. Since the functions $\overline{u}_i$ need not be the actual displacements of the material body, they are called *virtual displacements* and Equation 2.7 expresses the theorem of virtual work.

We have just proved that any equilibrium system satisfies relation 2.7 for every compatible system. The reverse is also true. Smooth functions $\tau_{ij} = \tau_{ji}$ satisfy the equilibrium equations 2.3 if and only if Equation 2.7 of virtual work holds for all smooth functions.

2.1.4 Constitutive Relations

For any stress analysis problem, we must supply a mathematical characterization of the material by specification of the relation between the stress and the strain. The classical theories of materials are elasticity, plasticity, and viscoelasticity. We will consider the theory of elasticity in this chapter. For elastic materials, the stress and strain tensors are related by

$$\tau_{ij} = \frac{\partial \mathcal{A}(\boldsymbol{\varepsilon})}{\partial \varepsilon_{ij}}, \tag{2.8}$$

where

$$\mathcal{A}(\boldsymbol{\varepsilon}) = \frac{1}{2} c_{ijkm} \varepsilon_{ij} \varepsilon_{km} + l_{ij} \varepsilon_{ij}. \tag{2.9}$$

The term $l_{ij} = l_{ji}$ is a function of temperature change and the constants c_{ijkm} satisfy the following symmetry conditions:

$$c_{ijkm} = c_{jikm} = c_{ijmk} = c_{kmij}. \tag{2.10}$$

Thus,

$$\tau_{ij} = l_{ij} + c_{ijkm} \varepsilon_{km}. \tag{2.11}$$

For an isotropic material with linear thermal expansion,

$$\mathcal{A}(\boldsymbol{\varepsilon}) = \mu \varepsilon_{km} \varepsilon_{km} + \frac{\lambda}{2} (\varepsilon_{kk})^2 - \beta T \varepsilon_{kk}, \tag{2.12}$$

and 2.11 simplifies to

$$\tau_{km} = -\beta T \delta_{km} + 2\mu \varepsilon_{km} + \lambda \varepsilon_{ii} \delta_{km}. \tag{2.13}$$

That is,

$$l_{ij} = -\beta T \delta_{ij}, \tag{2.14}$$

and

$$c_{ijkm} = \mu(\delta_{ik}\delta_{jm} + \delta_{im}\delta_{jk}) + \lambda\delta_{ij}\delta_{km}, \tag{2.15}$$

where β, μ, and λ are material constants and T is the temperature change. In terms of the shear modulus G, the modulus of elasticity E, the Poisson ratio ν, and the coefficient of linear thermal expansion α,

$$\begin{aligned} &\mu = G = \frac{E}{2(1+\nu)}, \quad \lambda = \frac{2\nu G}{1-2\nu}, \quad \beta = \frac{\alpha E}{1-2\nu}, \\ &\frac{\lambda}{\lambda+\mu} = 2\nu, \ 2\mu + 3\lambda = \frac{2\mu(1+\nu)}{1-2\nu}, \ \lambda + 2\mu = 2\nu. \end{aligned} \tag{2.16}$$

The constitutive relations for an isotropic material are simplified if we introduce the deviatoric components of the tensors, which are defined as follows:

$$\begin{aligned} s_{km} &= \tau_{km} - \frac{1}{3}\tau_{ii}\delta_{km}, \\ e_{km} &= \varepsilon_{km} - \frac{1}{3}\varepsilon_{ii}\delta_{km}. \end{aligned} \tag{2.17}$$

The constitutive relation 2.13 then becomes

$$\begin{aligned} s_{km} &= \mu 2 e_{km}, \\ \frac{1}{3}\tau_{kk} &= -\beta T + \kappa \varepsilon_{kk}, \end{aligned} \tag{2.18}$$

where κ is the bulk modulus,

$$\kappa = \lambda + \frac{2}{3}\mu = \frac{E}{3(1-2\nu)}. \tag{2.19}$$

The deviatoric components of strain are measures of distortion ($\gamma_{12} = 2e_{12}$ is the shear of the x–y fibers), and the first equation in 2.18 is therefore the constitutive relation for distortion. The quantity ε_{kk} is the volumetric strain, and the second equation in 2.18 is the constitutive relation for dilatation relating the mean stress $\tau_{kk}/3$ to the volumetric strain. The two relations in 2.18 are equivalent to the single equation 2.13. Constitutive relations for small deformations of isotropic materials that are not

elastic can be derived by replacing the simple proportionality in 2.18 with more complicated relations. We will consider two possibilities later when we study viscoelastic materials and the theory of plasticity.

2.1.5 Boundary Conditions and Initial Conditions

Now, consider a closed surface bounding all of the material. Let n_i denote the components of the unit vector normal to the surface and directed toward the exterior of the body. The force per unit area T_i exerted on the body at the surface by the surrounding material, or by the exterior world, is related to the stress tensor at a point on the surface by 2.6:

$$T_i = \tau_{ij} n_j. \tag{2.20}$$

The external surface of the body consists of two sets of points $\mathcal{S}_T$ and $\mathcal{S}_u$, where the stress vector or the displacement vector must be specified:

$$T_i = T_i^{\mathrm{o}} \quad \text{on} \quad \mathcal{S}_T, \tag{2.21}$$

$$u_i = u_i^{\mathrm{o}} \quad \text{on} \quad \mathcal{S}_u. \tag{2.22}$$

For dynamical problems, one must also specify initial conditions:

$$u_i \text{ and } \dot{u}_i \text{ given in } \mathcal{V} \text{ at } t = 0. \tag{2.23}$$

The exact solution to the elasticity problem for a given temperature is a set of functions τ_{ij}, ε_{ij}, and u_i satisfying the differential equations 2.1, 2.3, and 2.11, the boundary conditions 2.21 and 2.22, and the initial conditions 2.23.

2.1.6 Incompressible Materials

If the bulk modulus is very large compared to the shear modulus, a satisfactory solution of the elasticity equations can often be obtained by neglecting the volume change. The mean stress is then determined by the differential equations and the boundary conditions and not by the constitutive relations. The fundamental equations (no temperature change) are

$$\begin{aligned} &\varepsilon_{km} = \frac{1}{2}\left(\frac{\partial u_k}{\partial x_m} + \frac{\partial u_m}{\partial x_k}\right), \quad \varepsilon_{kk} = 0, \\ &\frac{\partial \tau_{ij}}{\partial x_j} + b_i = \rho \ddot{u}_i, \\ &\tau_{km} = -p\delta_{km} + 2\mu\varepsilon_{km} + \lambda\varepsilon_{ii}\delta_{km}. \end{aligned} \tag{2.24}$$

There is one additional equation ($\varepsilon_{kk} = 0$) and one additional unknown (p).

2.1.7 Plane Strain

There are two very important special cases in which the number of spatial coordinates that need to be considered can be reduced from three to two: plane stress and plane strain. For plane strain, $\varepsilon_{k3} = 0$ and all quantities are independent of the x_3 coordinate. Then, from 2.13,

$$\tau_{13} = 0, \quad \tau_{23} = 0, \quad \tau_{33} = -\beta T + \lambda \varepsilon_{\theta\theta}, \tag{2.25}$$

$$\tau_{\alpha\beta} = -\beta T \delta_{\alpha\beta} + 2\mu \varepsilon_{\alpha\beta} + \frac{2\nu\mu}{1-2\nu} \varepsilon_{\theta\theta} \delta_{\alpha\beta}. \tag{2.26}$$

From 2.1,

$$\varepsilon_{\alpha\beta} = \frac{1}{2}\left(\frac{\partial u_\alpha}{\partial x_\beta} + \frac{\partial u_\beta}{\partial x_\alpha} \right). \tag{2.27}$$

From 2.3,

$$\frac{\partial \tau_{\alpha\beta}}{\partial x_\beta} + b_\alpha = \rho \ddot{u}_\alpha, \tag{2.28}$$

since the derivatives with respect to x_3 are zero.

Equations 2.26–2.28 are a set of eight equations in eight unknowns (u_1, u_2, τ_{11}, τ_{22}, τ_{12}, ε_{11}, ε_{22}, ε_{12}) that are called the equations of plane strain for an isotropic material. Such relations can often be used to adequately characterize the deformations of cylindrical bodies with loads and temperature changes that do not vary along the length of the cylinder and where the ends of the cylinder are prevented from undergoing axial displacement ($\varepsilon_{33} \equiv 0$).

The constitutive relations 2.26 can be expressed in the form 2.8 if we redefine $\mathcal{A}$ and $l_{\alpha\beta}$ as follows.

$$\mathcal{A}(\boldsymbol{\varepsilon}) = \mu \varepsilon_{\alpha\beta} \varepsilon_{\alpha\beta} + \frac{\nu\mu}{1-2\nu} (\varepsilon_{\theta\theta})^2 + l_{\alpha\beta} \varepsilon_{\alpha\beta}, \tag{2.29}$$

with

$$l_{\alpha\beta} = \beta T \delta_{\alpha\beta} \tag{2.30}$$

for plane strain.

2.1.8 Plane Stress

For plane stress, $\tau_{k3} = 0$ and all quantities are independent of x_3. From 2.13, it follows that

$$\varepsilon_{33} = \frac{(1-2\nu)}{2\mu(1-\nu)} \beta T - \frac{\nu}{1-\nu} \varepsilon_{\theta\theta}, \tag{2.31}$$

$$\begin{aligned}\tau_{\alpha\beta} &= -\frac{(1-2\nu)}{1-\nu}\beta T\delta_{\alpha\beta} + 2\mu\varepsilon_{\alpha\beta} + \frac{2\mu\nu}{1-\nu}\varepsilon_{\theta\theta}\delta_{\alpha\beta} \\ &= -\frac{(1-2\nu)}{1-\nu}\beta T\delta_{\alpha\beta} + \left[\mu(\delta_{\alpha\theta}\delta_{\beta\phi} + \delta_{\alpha\phi}\delta_{\beta\theta}) + \frac{2\mu\nu}{1-\nu}\delta_{\alpha\beta}\delta_{\theta\phi}\right]\varepsilon_{\theta\phi}.\end{aligned} \tag{2.32}$$

From 2.1,

$$\varepsilon_{\alpha\beta} = \frac{1}{2}\left(\frac{\partial u_\alpha}{\partial x_\beta} + \frac{\partial u_\beta}{\partial x_\alpha}\right). \tag{2.33}$$

From 2.3,

$$\frac{\partial \tau_{\alpha\beta}}{\partial x_\beta} + b_\alpha = \rho\ddot{u}_\alpha. \tag{2.34}$$

Equations 2.32–2.34 are, again, a set of eight equations in eight unknowns that are called the equations of plane stress for an isotropic material. Such relations can often be used to adequately characterize the deformations of thin sheets that are loaded only on the edges.

The full equations of elasticity are only approximately satisfied by the plane stress assumptions because there is no solution to the geometric relations 2.1 for u_3 corresponding to the plane stress conditions. The stress and displacement should be viewed as the mean value over the thickness of the sheet. This is often referred to as *generalized plane stress*.[1]

The constitutive relations 2.32 can be expressed in form 2.8 if we redefine $\mathcal{A}$ and $l_{\alpha\beta}$ as follows.

$$\mathcal{A}(\boldsymbol{\varepsilon}) = \mu\varepsilon_{\alpha\beta}\varepsilon_{\alpha\beta} + \frac{\mu\nu}{1-\nu}(\varepsilon_{\theta\theta})^2 + l_{\alpha\beta}\varepsilon_{\alpha\beta}, \quad \text{(plane stress)} \tag{2.35}$$

$$l_{\alpha\beta} = -\frac{(1-2\nu)}{1-\nu}\beta T\delta_{\alpha\beta}. \quad \text{(plane stress)} \tag{2.36}$$

The equations of plane stress and plane strain have a formal similarity. Keeping μ the same, the plane strain equations 2.26 are transformed into the plane stress relations 2.32 if ν is replaced by $\nu/(1+\nu)$ and β by $\beta(1-2\nu)/(1-\nu)$. E is defined in each case by the values of ν and μ: $E = 2\mu(1+\nu)$,

2.1.9 Tensile Test

Let us consider the case when the body is a (possibly noncircular) cylinder and the x_1 axis is directed along the axis of the cylinder. At the ends $x_1 = 0$ and $x_1 = L$, there

is a uniform normal stress $\tau_{11} = p$ applied. The other surface tractions are zero. The material is isotropic and there is no temperature change. The solution to the elasticity equations is then

$$\begin{aligned} &\tau_{11} = p, \tau_{ij} = 0 \text{ otherwise,} \\ &\varepsilon_{11} = \frac{p}{E},\ \varepsilon_{22} = \varepsilon_{33} = -\nu\varepsilon_{11}, \\ &u_1(L) - u_1(0) = \frac{pL}{E}. \end{aligned} \tag{2.37}$$

We will use this as a test for the finite element program.

2.1.10 Pure Shear

A rectangular sheet of unit thickness is loaded by uniform shear stress along each edge (Figure 2.1). If the origin has zero displacement and the displacement $v = 0$ at point $(a, 0)$, the solution is

$$\begin{aligned} &\tau_{12} = p,\ \tau_{ij} = 0 \text{ otherwise,} \\ &\gamma_{12} = \frac{p}{G},\ \varepsilon_{11} = \varepsilon_{22} = 0, \\ &u = 0,\ v = \frac{p}{G}x. \end{aligned} \tag{2.38}$$

This can be used as a test problem for the finite element analysis.

2.1.11 Pure Bending

A rectangular sheet of unit thickness is loaded by linearly varying normal stress along each edge, Figure 2.2. If the origin has zero displacement and the displacement $v = 0$ at point $(a, 0)$, the solution is

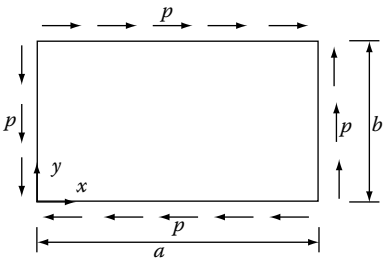


FIGURE 2.1 Pure shear.

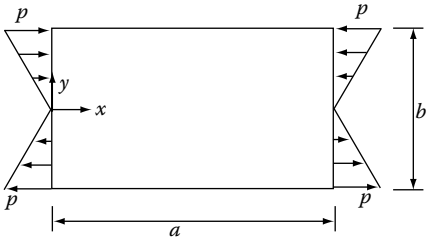


FIGURE 2.2 Pure bending.

$$\tau_{11} = -\frac{2p}{b}y,\ \tau_{ij} = 0 \text{ otherwise},$$

$$\varepsilon_{11} = -\frac{2p}{bE}y,\ \varepsilon_{22} = -\nu\varepsilon_{11},\ \gamma_{12} = 0, \tag{2.39}$$

$$u = +\frac{pa}{bE}y - \frac{2p}{bE}xy,\ v = -\frac{pa}{bE}x + \frac{p}{bE}x^2 + \frac{\nu p}{bE}y^2.$$

This can be used as a test problem for the finite element analysis.

2.1.12 Bending and Shearing

Consider the rectangular sheet shown in Figure 2.3. The body force and temperature change are zero. The top ($y = c$) and bottom ($y = -c$) edges are free of load. The right end ($x = 0$) has a shearing load of magnitude P that is parabolically distributed. The left end ($x = L$) has a parabolic shear load and a linearly varying normal traction:

$$x = 0:\ \tau_{xy} = \frac{3P}{4c}\left(1 - \frac{y^2}{c^2}\right).$$

$$x = L:\ \tau_{xy} = \frac{3P}{4c}\left(1 - \frac{y^2}{c^2}\right),\ \sigma_x = \frac{3PLy}{2c^3}. \tag{2.40}$$

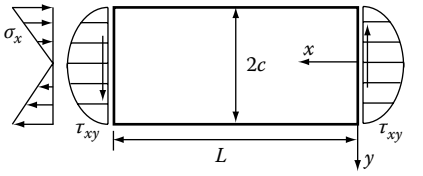


FIGURE 2.3 Bending by shear force.

An exact solution is readily found by expressing u and v as polynomials in x and y. For plane stress,

$$\begin{aligned}\sigma_x &= \frac{3Pxy}{2c^3},\\ \sigma_y &= 0,\\ \tau_{xy} &= \frac{3P}{4c}\left(1-\frac{y^2}{c^2}\right),\end{aligned} \tag{2.41}$$

and

$$\begin{aligned}u &= \frac{3Px^2y}{4c^3E}+\frac{\nu Py^3}{4c^3E}-\frac{Py^3}{4c^3G}+\frac{3Pyc^2}{8c^3G}+ey+g,\\ v &= -\nu\frac{3Pxy^2}{4c^3E}-\frac{Px^3}{4c^3E}+\frac{3Pxc^2}{8c^3G}-ex+h.\end{aligned} \tag{2.42}$$

The constants e, g, and h represent rigid translation and rotation, and have to be determined from appropriate support conditions. For the case $u(L,0) = 0$, $v(L,0) = 0$, $u(L,\pm c) = 0$,

$$\begin{aligned}g &= 0,\\ h &= -\frac{PL^3}{2c^3E}\left(1+\frac{4+5\nu}{2}\frac{c^2}{L^2}\right),\\ e &= -\frac{3PL^2}{4c^3E}-\frac{\nu Pc^2}{4c^3E}-\frac{P}{8cG}.\end{aligned} \tag{2.43}$$

The displacement at the origin is then

$$v(0,0) = -\frac{PL^3}{2c^3E}\left(1+\frac{4+5\nu}{2}\frac{c^2}{L^2}\right). \tag{2.44}$$

The first term is the displacement according to the elementary beam theory and the second is the additional displacement due to the shear that is neglected in the beam theory. For plane strain, replace ν by $\nu/(1-\nu)$ and E by $E/(1-\nu^2)$.

2.1.13 Properties of Solutions

Let us consider an equilibrium problem in the theory of elasticity for an isotropic material. The problem consists of determining the displacements, strains, and stresses that satisfy the partial differential equations of equilibrium,

$$\frac{\partial \tau_{ij}}{\partial x_j} + b_i = 0, \tag{2.45}$$

the constitutive relations,

$$\tau_{km} = 2\mu\varepsilon_{km} + \lambda\varepsilon_{ii}\delta_{km}, \tag{2.46}$$

the geometric equations,

$$\varepsilon_{km} = \frac{1}{2}\left(\frac{\partial u_k}{\partial x_m} + \frac{\partial u_m}{\partial x_k}\right), \tag{2.47}$$

the boundary conditions on load,

$$\tau_{ij} n_j = T_i^{\mathrm{o}} \quad \text{on} \quad \mathcal{S}_T, \tag{2.48}$$

and the support conditions,

$$u_i = u_i^{\mathrm{o}} \quad \text{on} \quad \mathcal{S}_u. \tag{2.49}$$

For plane stress or plane strain, there are corresponding equations in two dimensions.

Roughly speaking, a unique solution exists if the boundary of the body is smooth, the loads and support conditions are smooth, the support conditions prohibit rigid body displacements, and the material properties are smoothly distributed. The displacements, strains, and stresses are then continuous and finite throughout the body. However, problems for which there is a discontinuity in boundary, load, or support conditions are common. In such cases, the solution may have discontinuities or singularities.

One example of an isolated singularity is the apparently simple concept of a concentrated force. A concentrated force $\mathbf{F}$ applied at a point $\mathbf{x}_0$ on the external surface is mathematically characterized by the requirements:

$$\int\int_{S} \mathbf{T}\,\mathrm{d}A = \mathbf{F} \tag{2.50}$$

for all parts S of the surface that contain $\mathbf{x}_0$, and

$$\int\int_{S} \mathbf{T}\cdot\mathbf{u}(\mathbf{x})\,\mathrm{d}A = \mathbf{F}\cdot\mathbf{u}(\mathbf{x}_0) \tag{2.51}$$

for all fields **u**(**x**). The stress tensor becomes infinite at such points. Concentrated loads, of course, cannot occur in nature, so this is not considered a defect in the theory.

2.1.14 A Plane Stress Example with a Singularity in Stress

There are also many other apparently innocuous situations in which singularities occur. Consider, for instance, the plane stress example shown in Figure 2.4. We will use this problem repeatedly as a test problem for the finite element analysis and call it the *short beam problem.*

The exact solution to this simple-looking problem is unknown. In first approximation, one may apply the solution 2.41 with $P = pa$. In this case, shear stress τ_{12} on a cross section (y equals constant) varies parabolically with a maximum at the midpoint of the cross section:

$$\tau_{\max} = \frac{3}{2} p. \tag{2.52}$$

The normal stress, which varies linearly on each cross section with a maximum at the edge ($x = 0$, $x = a$), is given by

$$\sigma_{\max} = 6p\left(1 - \frac{y}{a}\right). \tag{2.53}$$

This approximation will certainly not be satisfactory near the support where the stress distribution depends strongly on the details of the support condition, nor near the top edge where the boundary condition is $\tau_{12} = p$. In fact, there is a discontinuity at each corner.

At the top corners, we have $\tau_{12} = p$ for $x = 0^+$ and $y = a$, whereas $\tau_{12} = 0$ for $x = 0$ and $y = a^-$. Therefore, the solution to this problem cannot be continuous at that corner.

At the bottom corners, the effect of the discontinuity in boundary conditions is less obvious. Let us consider the situation in the corner $x = 0$, $y = 0$, in more detail. One boundary condition along the edge $y = 0$ is $u_1(x,0) = 0$. By 2.47,

$$\varepsilon_{11}(x,0) = 0. \tag{2.54}$$

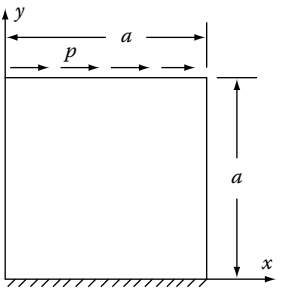


FIGURE 2.4 Short beam.

On the other hand, one boundary condition along the edge $x = 0$ is $\tau_{11} = 0$. From stress strain relation 2.46, we find that

$$\varepsilon_{11}(0, y) = -\nu \frac{\tau_{22}}{E} \neq 0. \tag{2.55}$$

From 2.54 and 2.55, we find the contradictory statements

$$\begin{aligned} &\varepsilon_{11}(0,0) = 0 \text{ for } x \to 0 \text{ along } y = 0, \\ &\varepsilon_{11}(0,0) \neq 0 \text{ for } y \to 0 \text{ along } x = 0. \end{aligned} \tag{2.56}$$

The mathematical consequence of this contradictory requirement is a singularity (infinite stress) at the corners where the free and fixed boundaries join.

The details of such a singularity are revealed by the solution of the problem for the infinite quarter plane ($x \geq 0$, $y \geq 0$) with zero displacements along the edge $y = 0$ and zero tractions along the edge $x = 0$. Using polar coordinates (r,θ), a solution has been found to be as follows[2]:

$$\begin{aligned} Eu_\theta &= r^{k+1}[(1+\nu)\alpha(A\cos\alpha\theta + B\sin\alpha\beta) \\ &\quad + (k(1+\nu) - 2(1-\nu))(C\cos k\theta + D\sin k\theta)]. \\ Eu_r &= r^{k+1}[(1+\nu)\alpha(A\sin\alpha\theta - B\cos\alpha\beta) \\ &\quad + (k(1+\nu) + 4)(C\sin k\theta - D\cos k\theta)]. \end{aligned} \tag{2.57}$$

$$\begin{aligned} \sigma_r &= r^k[(\alpha - \alpha^2)(A\cos\alpha\theta + B\sin\alpha\theta) \\ &\quad + (\alpha - k^2)(C\cos k\theta + D\sin k\theta)]. \\ \sigma_\theta &= r^k(\alpha^2 - \alpha)(A\cos\alpha\theta + B\sin\alpha\theta \\ &\quad + C\cos k\theta + D\sin k\theta). \\ \tau_{r\theta} &= r^k(\alpha - 1)[\alpha(A\sin\alpha\theta - B\cos\alpha\theta) \\ &\quad + k(C\sin k\theta - D\cos k\theta)]. \end{aligned} \tag{2.58}$$

The constant $\alpha = k + 2$. The constants A, B, C, and D are determined by the four boundary conditions on the edges $\theta = 0$ and $\theta = \pi/2$. The solution depends on the Poisson ratio. For $\nu = 0.32$, one finds a particular solution with $k = -1/4$. The displacements are finite at the corner ($r = 0$) but the stress components are infinite because of the negative exponent of r.

The numerical solutions that we will accomplish by the finite element method are approximations of the true solution to the field equations. The infinite stresses at a point are of course never found by numerical analysis, which deals only with finite real numbers. For each finer element mesh, one simply finds higher and higher stresses near the corners.

Another common situation where singularities occur is in the analysis of cracks that are modeled as a line or surface across which there is no contact force transmitted. In such problems, there is a singularity at the crack tip. A numerical method that introduces only continuous displacement fields cannot reproduce the exact solution. It is, however, possible to introduce special elements with built-in singularities in order to incorporate the crack tip singularities into the finite element model. This matter will be discussed later.

2.2 POTENTIAL ENERGY

Instead of seeking for the solution of the elasticity problem by solving the differential equations, we can seek those functions that make an appropriate operator stationary. First, let us consider the *potential energy* defined by

$$\mathcal{P} = \int_{\mathbf{V}} [\mathcal{A}(\boldsymbol{\varepsilon}) - b_k u_k] \mathrm{d}V - \int_{\mathbf{S}_T} T_k^0 u_k \, \mathrm{d}A, \tag{2.59}$$

where $\mathbf{S}_T$ is the part of the boundary on which the traction vector has the given value T_k^0. The strain tensor is defined by the kinematical relations 2.1, which is repeated here:

$$\varepsilon_{km} = u(k,m). \tag{2.60}$$

The stress tensor is regarded as defined by 2.8, which is repeated here:

$$\tau_{km} = \frac{\partial \mathcal{A}}{\partial \varepsilon_{km}}. \tag{2.61}$$

The potential energy is defined for all continuous functions u_k with piecewise continuous first derivatives and which satisfy the boundary conditions on displacement:

$$u_k = u_k^{\mathrm{o}} \quad \text{on} \quad \mathcal{S}_u. \tag{2.62}$$

For each set of functions u_k, the integrals over the given region can be evaluated for a given material and a numerical value for $\mathcal{P}$ obtained. Imagine that this is done for all possible admissible functions and the values of $\mathcal{P}$ so obtained are compared. One would find that the true solution is the one that gives $\mathcal{P}$ a minimum value.

To prove this assertion, suppose that vector $\mathbf{u}$ denotes the displacement vector for which $\mathcal{P}$ is a minimum. A necessary condition that $\mathcal{P}$ is a minimum is that the derivative of $\mathcal{P}$ is zero at point u. This condition is equivalent to the requirement that, for all vectors $\bar{\mathbf{u}}$,

$$\left.\frac{\partial \mathcal{P}(\mathbf{u}+\alpha\bar{\mathbf{u}})}{\partial \alpha}\right|_{\alpha=0} = 0. \tag{2.63}$$

In carrying out this calculation for $\mathcal{P}$ defined by 2.59, note that

$$\mathcal{A}(\mathbf{u}+\alpha\bar{\mathbf{u}}) = \mathcal{A}(\boldsymbol{\varepsilon}+\alpha\bar{\boldsymbol{\varepsilon}}), \tag{2.64}$$

where

$$\bar{\varepsilon}_{km} = \bar{u}_{(k,m)}. \tag{2.65}$$

Therefore,

$$\left.\frac{\partial \mathcal{A}(\boldsymbol{\varepsilon}+\alpha\bar{\boldsymbol{\varepsilon}})}{\partial \alpha}\right|_{\alpha=0} = \frac{\partial \mathcal{A}}{\partial \varepsilon_{km}}\bar{\varepsilon}_{km}. \tag{2.66}$$

Then, one obtains from 2.63 the following condition

$$\int_{\mathcal{V}} \{\tau_{km}\bar{u}_{(k,m)} - b_k\bar{u}_k\}\mathrm{d}V - \int_{S_T} T_k^0\bar{u}_k\mathrm{d}A = 0. \tag{2.67}$$

Note that 2.67 is identical to the virtual work expression for the compatible system $(\bar{u}_i, \bar{\varepsilon}_{ij})$ and $\bar{u}_i = 0$ on S_u. Upon integrating the first term by parts, one finds

$$-\int_{\mathcal{V}} (\tau_{km,m} + b_k)\bar{u}_k \,\mathrm{d}V + \int_{S_T} (\tau_{km}n_m - T_k^0)\bar{u}_k \,\mathrm{d}A = 0. \tag{2.68}$$

Since this relation must hold for arbitrarily selected values of $\bar{\mathbf{u}}$, each of the expressions in parentheses must be zero in the indicated region. Thus, the condition that the first derivative of $\mathcal{P}$ is a zero at $\mathbf{u}$, together with 2.2 and 2.3, is equivalent to the requirement that the differential equations and boundary conditions 2.45 to 2.49 are satisfied.

The procedure used in this section is called the "calculus of variations" in the older literature. It developed before the modern concepts of abstract operator and

abstract derivative. Calculation of the functional derivation of $\mathcal{P}$ is equivalent to calculation of "the first variation of $\mathcal{P}$."

We have not shown that $\mathcal{P}$ is a minimum at the true solution. The preceding calculations only show that the first derivative is zero at the true solution and vice versa. This is a necessary condition for a minimum, but it is not sufficient. Thus far, one can only say that the potential energy is *stationary* at the true solution, and it might be a maximum or a minimum or something like a point of inflection.

2.2.1 Proof of Minimum Potential Energy

We will now prove that the potential energy is indeed a minimum at the true solution in the case of isothermal deformations. The proof hinges on the following property of the strain energy $\mathcal{A}$. It is easy to see for the isotropic material that $\mathcal{A}(\boldsymbol{\varepsilon})$ is nonnegative for every tensor $\boldsymbol{\varepsilon}$ since it is the sum of squares of the components of $\boldsymbol{\varepsilon}$. It is zero only when all components of the strain tensor are zero. We assume that the strain energy always has this property.

Let u_i^* be any set of smooth functions that satisfy the displacement boundary conditions:

$$u_i^* = u_i^o \quad \text{on} \quad S_u. \tag{2.69}$$

Let ε_{ij}^* be the set of functions derived from u_i^* by the strain–displacement equations:

$$\varepsilon_{ij}^* = u_{(i,j)}^*. \tag{2.70}$$

Let τ_{ij}^* be the set of functions derived from ε_{ij}^* by the constitutive relations:

$$\tau_{ij}^* = c_{ijkm}\varepsilon_{km}^*. \tag{2.71}$$

These stresses need not satisfy the equilibrium equations or the boundary conditions on load. If those relations are satisfied, then the system ($\mathbf{u}^*$, $\boldsymbol{\varepsilon}^*$, $\boldsymbol{\tau}^*$) is the actual solution. We wish to consider the case when it is not the true solution.

Suppose the actual solution to the given boundary value problem is denoted by $\mathbf{u}$, $\boldsymbol{\varepsilon}$, $\boldsymbol{\tau}$. We first note that

$$\tau_{ij}^*\varepsilon_{ij} = c_{ijkm}\varepsilon_{km}^*\varepsilon_{ij} = c_{kmij}\varepsilon_{km}^*\varepsilon_{ij} = \tau_{km}\varepsilon_{km}^* \tag{2.72}$$

because of the symmetry of c_{ijkm}. Now, consider the expression

$$\frac{1}{2}(\tau_{ij}^* - \tau_{ij})(\varepsilon_{ij}^* - \varepsilon_{ij}) = \frac{1}{2}c_{ijkm}(\varepsilon_{ij}^* - \varepsilon_{ij})(\varepsilon_{km}^* - \varepsilon_{km}) = \mathcal{A}(\boldsymbol{\varepsilon}^* - \boldsymbol{\varepsilon}) > 0. \tag{2.73}$$

Using relations 2.72 and 2.73, it follows that

$$\begin{aligned} 2(\tau_{ij}^* - \tau_{ij})\varepsilon_{ij} &= \tau_{ij}^*\varepsilon_{ij}^* - \tau_{ij}\varepsilon_{ij} - \tau_{ij}^*(\varepsilon_{ij}^* - \varepsilon_{ij}) - \varepsilon_{ij}(\tau_{ij} - \tau_{ij}^*) \\ &= \tau_{ij}^*\varepsilon_{ij}^* - \tau_{ij}\varepsilon_{ij} - (\tau_{ij}^* - \tau_{ij})(\varepsilon_{ij}^* - \varepsilon_{ij}) \\ &< \tau_{ij}^*\varepsilon_{ij}^* - \tau_{ij}\varepsilon_{ij}. \end{aligned} \tag{2.74}$$

Consequently, using 2.72,

$$\frac{1}{2}(\tau_{ij}^*\varepsilon_{ij}^* - \tau_{ij}\varepsilon_{ij}) > \tau_{km}(\varepsilon_{km}^* - \varepsilon_{km}). \tag{2.75}$$

By the virtual work formula,

$$\int (\varepsilon_{ij}^* - \varepsilon_{ij})\tau_{ij}\,\mathrm{d}V = \int_{S_\mathrm{F}} (u_i^* - u_i)T_i^0\,\mathrm{d}A + \int_V (u_i^* - u_i)b_i\,\mathrm{d}V, \tag{2.76}$$

since $u_i = u_i^* = u_i^0$ on S_u. Integrating expression 2.75 over the body, using 2.76, and rearranging the terms, we find

$$\frac{1}{2}\int_V \tau_{ij}^*\varepsilon_{ij}^*\,\mathrm{d}V - \int_{S_\mathrm{F}} u_i^*T_i^0\,\mathrm{d}A - \int_V u_i^*b_i\,\mathrm{d}V > \frac{1}{2}\int \tau_{ij}\varepsilon_{ij}\,\mathrm{d}V - \int_{S_\mathrm{F}} u_iT_i^0\,\mathrm{d}A - \int_V u_ib_i\,\mathrm{d}V. \tag{2.77}$$

Thus, the potential energy of any compatible system is greater than the potential energy of the actual solution:

$$\mathcal{P}(\mathbf{u}^*) > \mathcal{P}(\mathbf{u}). \tag{2.78}$$

That is, the potential energy is a minimum for the actual solution. Among all the displacement fields that satisfy the support conditions, the solution for the given load condition is the displacement field that makes the potential energy a minimum.

2.3 MATRIX NOTATION

We will often use matrix notation instead of the component notation. Each notation has certain advantages. In matrix notation, the components of the stress and strain tensors will be arranged in a column matrix. For the plane problems,

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ 2\varepsilon_{12} \end{bmatrix}, \quad \boldsymbol{\tau} = \begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{12} \end{bmatrix}. \tag{2.79}$$

For three-dimensional problems,

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{12} \\ 2\varepsilon_{23} \\ 2\varepsilon_{31} \end{bmatrix}, \quad \boldsymbol{\tau} = \begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{bmatrix}. \tag{2.80}$$

Note that this notation is not the usual one for tensor analysis where the components of second-order tensors are arranged in a square matrix.

In matrix notation, the constitutive relations for three-dimensional problems and plane problems both have the form

$$\boldsymbol{\tau} = \boldsymbol{l} + \mathbf{C}\boldsymbol{\varepsilon}, \tag{2.81}$$

with appropriate definitions of $\boldsymbol{l}$ and $\mathbf{C}$. For three-dimensional problems,

$$\mathbf{C} = \begin{bmatrix} 2\mu+\lambda & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & 2\mu+\lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & 2\mu+\lambda & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}, \quad \boldsymbol{l} = -\beta T \begin{bmatrix} 1 \\ 1 \\ 1 \\ 0 \\ 0 \\ 0 \end{bmatrix}. \tag{2.82}$$

For plane strain,

$$\mathbf{C} = \begin{bmatrix} 2\mu+\lambda & \lambda & 0 \\ \lambda & 2\mu+\lambda & 0 \\ 0 & 0 & \mu \end{bmatrix}, \quad \boldsymbol{l} = -\beta T \begin{bmatrix} 1 \\ 1 \\ 0 \end{bmatrix}, \quad 2\mu+\lambda = \frac{2\mu(1-\nu)}{(1-2\nu)}. \tag{2.83}$$

For plane stress,

$$\mathbf{C} = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix}, \quad \boldsymbol{l} = -\beta T \frac{1-2\nu}{1-\nu}\begin{bmatrix} 1 \\ 1 \\ 0 \end{bmatrix}. \tag{2.84}$$

In practical applications, materials are usually orthotropic at most and data are usually stated for the inverse of $\mathbf{C}{:}\boldsymbol{\varepsilon} = \mathbf{C}^{-1}\boldsymbol{\tau}$. For orthotropic materials, $\mathbf{C}^{-1}$ has the form:

$$\mathbf{C}^{-1} = \begin{bmatrix} \dfrac{1}{E_x} & -\dfrac{\nu_{xy}}{E_y} & -\dfrac{\nu_{xz}}{E_z} & 0 & 0 & 0 \\ -\dfrac{\nu_{yx}}{E_x} & \dfrac{1}{E_y} & -\dfrac{\nu_{yz}}{E_z} & 0 & 0 & 0 \\ -\dfrac{\nu_{zx}}{E_x} & -\dfrac{\nu_{zy}}{E_y} & \dfrac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \dfrac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \dfrac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \dfrac{1}{G_{xz}} \end{bmatrix}, \quad \begin{aligned} \frac{\nu_{xy}}{E_y} &= \frac{\nu_{yx}}{E_x} \\ \frac{\nu_{xz}}{E_z} &= \frac{\nu_{zx}}{E_x} \\ \frac{\nu_{yz}}{E_z} &= \frac{\nu_{zy}}{E_y} \end{aligned} . \tag{2.85}$$

The components of the vectors T_i, b_i, and u_i are arranged in column matrices in the usual manner: the index indicates the row number. The virtual work formula then takes the form:

$$\int_{\mathrm{S}} \bar{\mathbf{u}}^{\mathrm{T}}\mathbf{T}\,\mathrm{d}A + \int_{\mathrm{V}} \bar{\mathbf{u}}^{\mathrm{T}}\mathbf{b}\,\mathrm{d}V = \int_{\mathrm{V}} \bar{\boldsymbol{\varepsilon}}^{\mathrm{T}}\boldsymbol{\tau}\,\mathrm{d}V. \tag{2.86}$$

For isothermal deformations, the strain energy becomes

$$\mathcal{A}(\boldsymbol{\varepsilon}) = \frac{1}{2}c_{ijkm}\varepsilon_{ij}\varepsilon_{km} = \frac{1}{2}\tau_{ij}\varepsilon_{ij} = \frac{1}{2}\boldsymbol{\tau}^{\mathrm{T}}\boldsymbol{\varepsilon}. \tag{2.87}$$

The potential energy then becomes

$$\mathcal{P} = \int_{\mathcal{V}}\left[\frac{1}{2}\boldsymbol{\tau}^{\mathrm{T}}\boldsymbol{\varepsilon} - \mathbf{b}^{\mathrm{T}}\mathbf{u}\right]\mathrm{d}V - \int_{\mathcal{S}_T} (\mathbf{T}_0)^{\mathrm{T}}\mathbf{u}\,\mathrm{d}A. \tag{2.88}$$

The form for plane problems is analogous.

2.4 AXIALLY SYMMETRIC DEFORMATIONS

2.4.1 Cylindrical Coordinates

An important special case of the general deformations occurs when the displacements are symmetrical with respect to a line that is generally chosen as the z axis. To analyze the axisymmetric deformations, we introduce a cylindrical coordinate system (Figure 2.5). The components of the displacement vector, the stress tensor, and the strain tensor are represented by components with respect to unit vectors along the coordinate lines (Figures 2.5 and 2.6).

The momentum equations 2.3 become

$$\begin{aligned}
&\frac{\partial \tau_{rr}}{\partial r}+\frac{1}{r}\frac{\partial \tau_{\theta r}}{\partial \theta}+\frac{\partial \tau_{zr}}{\partial z}+\frac{\tau_{rr}-\tau_{\theta\theta}}{r}+\rho b_r=\rho a_r,\\
&\frac{\partial \tau_{r\theta}}{\partial r}+\frac{1}{r}\frac{\partial \tau_{\theta\theta}}{\partial \theta}+\frac{\partial \tau_{z\theta}}{\partial z}+\frac{\tau_{r\theta}+\tau_{\theta r}}{r}+\rho b_\theta=\rho a_\theta,\\
&\frac{\partial \tau_{rz}}{\partial r}+\frac{1}{r}\frac{\partial \tau_{\theta z}}{\partial \theta}+\frac{\partial \tau_{zz}}{\partial z}+\frac{\tau_{rz}}{r}+\rho b_z=\rho a_z.
\end{aligned} \tag{2.89}$$

The strain–displacement relations 2.1 become

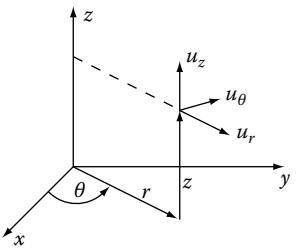


FIGURE 2.5 Displacements in cylindrical coordinates.

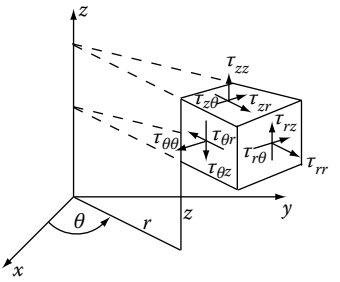


FIGURE 2.6 Stress components in cylindrical coordinates.

$$\varepsilon_{rr} = \frac{\partial u_r}{\partial r}, \quad \varepsilon_{\theta\theta} = \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{u_r}{r}, \quad \varepsilon_{zz} = \frac{\partial u_z}{\partial z},$$
$$\gamma_{r\theta} = \frac{1}{r}\frac{\partial u_r}{\partial \theta} + \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r}, \quad \gamma_{\theta z} = \frac{1}{r}\frac{\partial u_z}{\partial \theta} + \frac{\partial u_\theta}{\partial z}, \quad \gamma_{zr} = \frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z}. \tag{2.90}$$

The stress–strain relations for an isotropic material are 2.3 with

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{rr} \\ \tau_{\theta\theta} \\ \tau_{zz} \\ \tau_{r\theta} \\ \tau_{\theta z} \\ \tau_{zr} \end{bmatrix}, \quad \boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{zz} \\ \gamma_{r\theta} \\ \gamma_{\theta z} \\ \gamma_{zr} \end{bmatrix}. \tag{2.91}$$

2.4.2 Axial Symmetry

The three-dimensional equations are of limited use in numerical analysis, where we will normally use the global Cartesian coordinates. However, when deformations are symmetrical about the z axis so that $u_\theta = 0$ and all quantities are independent of θ, a two-dimensional problem occurs where $\gamma_{\theta z} = 0$, $\gamma_{r\theta} = 0$, and all quantities depend only on (r,z). The remaining geometric relations from 2.89 are

$$\varepsilon_{rr} = \frac{\partial u_r}{\partial r}, \quad \varepsilon_{\theta\theta} = \frac{u_r}{r}, \quad \varepsilon_{zz} = \frac{\partial u_z}{\partial z}, \quad \gamma_{zr} = \frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z}. \tag{2.92}$$

The stress–strain relations 2.91 show that $\tau_{\theta z} = 0$, $\tau_{r\theta} = 0$, and

$$\begin{aligned} \tau_{rr} &= -\beta T + 2\mu\varepsilon_{rr} + \lambda\left(\varepsilon_{rr} + \varepsilon_{\theta\theta} + \varepsilon_{zz}\right), \\ \tau_{\theta\theta} &= -\beta T + 2\mu\varepsilon_{\theta\theta} + \lambda\left(\varepsilon_{rr} + \varepsilon_{\theta\theta} + \varepsilon_{zz}\right), \\ \tau_{zz} &= -\beta T + 2\mu\varepsilon_{zz} + \lambda\left(\varepsilon_{rr} + \varepsilon_{\theta\theta} + \varepsilon_{zz}\right), \\ \tau_{zr} &= \mu\gamma_{zr}. \end{aligned} \tag{2.93}$$

The momentum equations 2.89 reduce to

$$\begin{aligned} &\frac{\partial \tau_{rr}}{\partial r} + \frac{\partial \tau_{zr}}{\partial z} + \frac{\tau_{rr} - \tau_{\theta\theta}}{r} + \rho b_r = \rho a_r, \\ &\frac{\partial \tau_{rz}}{\partial r} + + \frac{\partial \tau_{zz}}{\partial z} + \frac{\tau_{rz}}{r} + \rho b_z = \rho a_z. \end{aligned} \tag{2.94}$$

The axisymmetric equations are usually applied to a circular cylinder with the generator along the z axis and surface loads and supports symmetrical about the z axis.

2.4.3 Plane Stress and Plane Strain

The cylindrical coordinates are often useful for the analytical formulation and solution of plane problems where the region is a cross section of a circular cylinder. In this case, $\tau_{zr} = 0$, $\tau_{z\theta} = 0$, and all quantities depend only on (r,θ). The balance of momentum 2.89 gives

$$\begin{aligned}\frac{\partial \tau_{rr}}{\partial r} + \frac{1}{r}\frac{\partial \tau_{\theta r}}{\partial \theta} + \frac{\tau_{rr} - \tau_{\theta\theta}}{r} + \rho b_r &= \rho a_r,\\ \frac{\partial \tau_{r\theta}}{\partial r} + \frac{1}{r}\frac{\partial \tau_{\theta\theta}}{\partial \theta} + \frac{\tau_{r\theta} + \tau_{\theta r}}{r} + \rho b_\theta &= \rho a_\theta.\end{aligned} \tag{2.95}$$

The geometric relations 2.90 give

$$\varepsilon_{rr} = \frac{\partial u_r}{\partial r}, \quad \varepsilon_{\theta\theta} = \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{u_r}{r}, \quad \gamma_{r\theta} = \frac{1}{r}\frac{\partial u_r}{\partial \theta} + \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r}. \tag{2.96}$$

The stress–strain relations for an isotropic material are 2.83 and 2.84 with

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{rr} \\ \tau_{\theta\theta} \\ \tau_{r\theta} \end{bmatrix}, \quad \boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \gamma_{r\theta} \end{bmatrix}. \tag{2.97}$$

2.5 PROBLEMS

1. For an isotropic material, prove that $s_{km} = \mu 2 e_{km}$ and $\frac{1}{3}\tau_{kk} = -\beta T + B\,\varepsilon_{kk}$.
2. For an isotropic material, prove that $\varepsilon_{km} = \frac{1+\nu}{E}\tau_{km} - \frac{\nu}{E}\tau_{ii}\delta_{km} + \alpha T \delta_{km}$.
3. Determine the substitution for ν that transforms the plane stress equations into plane strain equations.
4. For an isotropic material, (a) calculate the components of the strain tensor and the stress tensor for the following set of given displacements for an isotropic material:

$$u_1 = \frac{\sigma}{E}x_1, \quad u_2 = -\nu\frac{\sigma}{E}x_2, \quad u_3 = -\nu\frac{\sigma}{E}x_3,$$

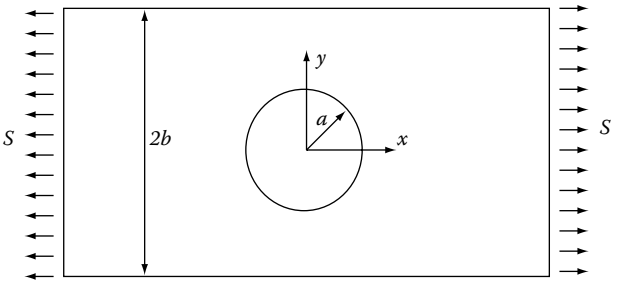


FIGURE 2.7 Sheet with a hole.

where σ is a constant. (b) Check the equilibrium equations to see if they are satisfied for zero body forces. (c) Show the edge tractions on a diagram of the body $0 \le x_1 \le L$, $0 \le x_2 \le a$, $0 \le x_3 \le b$.

5. A rectangular sheet of unit thickness is loaded by uniform shear stress along each edge (see Figure 2.1). If the origin has zero displacement and the displacement $v = 0$ at point $(a, 0)$, the proposed solution is given by 2.38. Check the equations of linear elasticity for an isotropic material to see if this proposed solution is correct for zero body forces and zero acceleration.
6. Consider the plane stress problem diagrammed in Figure 2.2. The origin has zero displacement and $v = 0$ at point $(a,0)$. Check the proposed solution 2.39 for zero body force and zero inertial force.
7. The plane stress problem of Figure 2.4 is to be analyzed using only the left half $(0 \le x \ge a/2)$ and symmetry/antisymmetry. Determine the boundary conditions to be applied to the line $x = a/2$.
8. The plane stress problem in Figure 2.3 is to be analyzed using only the upper half $(0 \le y \ge c)$ and symmetry/antisymmetry. Determine the boundary conditions to be applied to line $y = 0$.
9. The plane stress problem of stretching of a sheet with a hole shown in Figure 2.7 is to be solved using only the upper right quarter. Determine the boundary conditions to be applied to the lines $x = 0$ and $y = 0$.
10. A solid cylinder of radius R and length L is loaded by a uniformly distributed tensile load on each end: $\tau_{zz} = \sigma$ for $z = 0$ and $z = L$. Assume for a trial solution that the deformations are axisymmetric and $u_z = c_1 z$ and $u_r = c_2 r$, where c_1 and c_2 are constants. Show that the elasticity equations and boundary conditions on load can be satisfied and determine the required values of constants c_1 and c_2.

REFERENCES

1. Sololnikoff, I. S., *Mathematical Theory of Elasticity,* McGraw-Hill, New York, 1956, 67.
2. Williams, M. L., Stress singularities resulting from various boundary conditions in angular corners of plates in extension, *J. Appl. Mech.* 19, 526, 1952.

BIBLIOGRAPHY

Dill, E. H., *Continuum Mechanics: Elasticity, Plasticity, Viscoelasticity*, CRC Press, Boca Raton, FL, 2007.

Malvern, L. E., *Introduction to the Mechanics of Continuous Media*, Prentice-Hall, Englewood Cliffs, NJ, 1969.

Sololnikoff, I. S., *Mathematical Theory of Elasticity,* McGraw-Hill, New York, 1956.

Timoshenko, S., and J. N. Goodier, *Theory of Elasticity*, McGraw-Hill, New York, 1951.

Washizu, K., *Variational methods in elasticity and plasticity*, 2nd ed., Pergamon Press, New York, 1974.

3 Finite Element Method for Linear Elasticity

A linear elastic analysis is the basis of most engineering designs. However, exact solutions to the differential equations of linear elasticity are known for only the simplest loadings on the simplest configurations. One needs a way to construct solutions for any configuration and all load and support conditions. The finite element method is well suited for this purpose. We are concerned with a procedure for obtaining an accurate numerical solution to the exact equations.

However, the engineer should not forget that the theory of linear elasticity is an imperfect model of material behavior. Its usefulness in engineering design is limited by the simplicity of the model, and because the exact mathematical solutions of boundary value problems are sometimes physically unrealistic. Consequently, the effort to obtain extremely accurate numerical solutions may not be worthwhile in some cases. The limits on applicability of the model to the physical situation must be kept in mind when the solution of the elasticity problem is used in an engineering application.

The finite element method was developed in detail in Chapter 1 for naturally articulated structures. We now investigate the method for two- and three-dimensional solids. The fundamental idea is to divide the body into small pieces that we call *elements*. The displacement field is then approximated in each element by interpolating between the values of the displacement at specific points on the element that we call *nodes*. The displacement field generated in this manner is substituted into the expression for the potential energy, which then depends only on the nodal displacements. The condition that $\mathcal{P}$ is a minimum generates a set of linear algebraic equations for the nodal displacements.

In the derivation of the theorem of minimum potential energy in Chapter 2, we assumed that the displacement field is continuous. We should therefore choose the expressions for the displacements within an element in such a way that the global displacement field is continuous for any and all nodal displacements. If this is done, the elements are said to be *conforming*. Nonconforming elements may be satisfactory in certain cases, which we will discuss later.

We number the elements consecutively from 1, 2, 3, …, M and also number the components of unknown nodal displacement throughout the body consecutively from 1, 2, 3, …, N. For three dimensions, N is three times the number of nodes. For plane stress or plane strain, N is two times the number of nodes.

In the following discussion of the finite element method, we will assume that the boundary of the body coincides with the boundary of the finite element model. This eliminates the geometric error resulting from the approximation for curved boundaries by polygonal elements. In practical applications, that is often not the case and

there will be a geometric error; but, of course, we suppose that error tends to zero as the size of the elements tends to zero.

3.1 FINITE ELEMENT APPROXIMATION

Let us consider one particular element and one component u_i of displacement within that element. The component u_i will depend on the nodal displacements D_K for that element, but only on the displacements at nodes falling within that element or on its boundary, and not on the displacement of a node that does not fall within that element. We will only use an interpolation process for which the relation is linear:

$$u_i(\mathbf{x}) = \sum_{K \in I_m} N_{iK}(\mathbf{x}) D_K, \tag{3.1}$$

where I_m is the set of nodal displacements for the mth element. In this chapter, we will apply the summation convention only to the lower-case indices. The index i has the range (1, 2) for the plane problem and the range (1, 2, 3) for the three-dimensional problem. The interpolation functions N_{iK} are called *shape functions* for the element. Different shape functions could be used for each element but we will usually use the same ones for all elements. Furthermore, each component of displacement will depend only on the nodal values of that same component.

The shape functions must be such that a continuous displacement field is generated that is capable of approximating the true solution with an error that tends to zero as the size of each element tends to zero. In particular, we must have the following conditions:

(1) Continuity along element boundaries for arbitrary nodal displacements.
(2) Shape functions must be able to exactly represent a state of constant strain.
(3) If $u_i(\mathbf{x}_J) = D_M$, then

$$\sum_{K \in I_m} N_{iK}(\mathbf{x}_J) D_K = D_M. \tag{3.2}$$

The second condition will be satisfied if a polynomial is used that contains the constant and linear terms. We will consider the choice of shape functions in more detail in subsequent chapters.

Given the shape functions, the strains are calculated by Equation 2.1:

$$\varepsilon_{ij} = \sum_{K \in I_m} A_{ijK} D_K, \tag{3.3}$$

where

$$A_{ijK} = \frac{1}{2}\left(\frac{\partial N_{iK}}{\partial x_j} + \frac{\partial N_{jK}}{\partial x_i} \right). \tag{3.4}$$

The stress, calculated by 2.11, has the form

$$\tau_{ij} = l_{ij} + \sum_{K \in I_m} B_{ijK} D_K, \tag{3.5}$$

where

$$B_{ijK} = c_{ijkm} A_{kmK}. \tag{3.6}$$

The values of c_{ijkm} and l_{ij} depend on whether the problem is plane stress or plane strain or three dimensional. From 2.9, for each element

$$\begin{aligned} \mathcal{A}(\boldsymbol{\varepsilon}) &= \frac{1}{2} c_{ijkm} \varepsilon_{ij} \varepsilon_{km} + l_{ij} \varepsilon_{ij} \\ &= \sum_{K \in I_m} \sum_{M \in I_m} \frac{1}{2} A_{ijK} B_{ijM} D_K D_M + \sum_{K \in I_m} l_{ij} A_{ijK} D_K. \end{aligned} \tag{3.7}$$

Indices K and M range over the index set for the particular element.

3.1.1 Potential Energy

The elements partition the region $\mathcal{V}$ into subregions $\mathcal{V}_m$. Let $\mathcal{S}_m$ denote the part, if any, of the boundary of the subregion $\mathcal{V}_m$ that lies on the exterior surface. We will initially regard the entire exterior surface as having specified loads. Then, the potential energy 2.59 is the sum of the potential energy of each element:

$$\mathcal{P} = \sum_{m=1}^{M} \mathcal{P}_m, \tag{3.8}$$

and

$$\mathcal{P}_m = \int_{\mathcal{V}_m} [\mathcal{A}(\boldsymbol{\varepsilon}) - b_i u_i] \, \mathrm{d}V - \int_{\mathcal{S}_m} T_i^{\mathrm{o}} u_i \, \mathrm{d}A. \tag{3.9}$$

For plane stress or plane strain, the volume integrals reduce to integrals over the two-dimensional region of the element and the bounding surface of the body is a curve. Next, substitute expression 3.1 for the displacement field of an element, and expression 3.7 for the strain energy, into expression 3.9 for the potential energy of the element. This gives

$$\mathcal{P}_m = \sum_{I,J \in I_m} \frac{1}{2} k_{IJ}^m D_I D_J - \sum_{I \in I_m} f_I^m D_I, \tag{3.10}$$

where

$$k_{IJ}^{m} = \int_{\mathcal{V}_m} B_{kiI} A_{kiJ} \, \mathrm{d}V, \tag{3.11}$$

$$f_I^m = -\int_{\mathbf{V}_m} l_{ki} A_{kiI} \, \mathrm{d}V + \int_{\mathbf{V}_m} b_k N_{kI} \, \mathrm{d}V + \int_{\mathbf{S}_m} T_k^{\circ} N_{kI} \, \mathrm{d}A. \tag{3.12}$$

Expression 3.8 for the total potential energy of the body is now a summation of the corresponding terms in 3.10. All of the coefficients of a particular pair, D_I,D_J, I,J = 1 to N, are added together. If a pair does not appear in 3.10, the coefficient is zero. Thus,

$$\mathcal{P} = \sum_{I=1}^{N} \sum_{J=1}^{N} \frac{1}{2} K_{IJ} D_I D_J - \sum_{J=1}^{N} F_J D_J, \tag{3.13}$$

where

$$K_{IJ} = \sum_{m=1}^{M} k_{IJ}^{m}, \tag{3.14}$$

$$F_I = \sum_{m=1}^{M} f_I^m. \tag{3.15}$$

For three dimensions, the range N of (I,J) is from one to three times the number of nodes. For the plane problem, the sum is twice the number of nodes. The matrix of elements K_{IJ} is called the *global stiffness matrix*. The matrix with elements F_I is called the *global force matrix*. The matrix $\mathbf{K}$ has N rows and columns. The matrix $\mathbf{F}$ has N rows. The total number of displacement components N is called the *degree of freedom* of the unsupported body.

The summation in 3.14 and 3.15 is in the sense of a merge of the element matrices as detailed in Chapter 1. Since the range of I and J in 3.10 depends on the element m, we can view the merge in two ways.

First, in order to carry out the sum on m, we can extend the summation in 3.10 to all displacement parameters by defining k_{IJ}^m to be zero for all I and J that are not in the set I_m. Then the merge of element matrices is a simple addition of all element matrices with like indices. This is the method of augmented matrices detailed in Chapter 1. Alternatively, we can set $K_{IJ} = 0$ initially, and then, for each element, we add k_{IJ}^m to the K_{IJ} term with matching indices. This is the process of direct merge of the element matrices.

The stiffness matrix for the unsupported structure has a zero determinant since the displacements can be determined only to within a rigid translation and rotation.

The components of the force matrix satisfy the identity

$$\sum_J F_J D_J \equiv \int_S T_i u_i \, \mathrm{d}A + \int_{\mathcal{V}} b_i u_i \, \mathrm{d}V - \int_{\mathcal{V}} l_{ij} \varepsilon_{ij} \, \mathrm{d}V. \tag{3.16}$$

That is, each F_I is energy equivalent to a concentrated force corresponding to the displacement component D_I. Loads satisfying 3.16 are called *consistent* with the energy formulation.

3.1.2 Finite Element Equations

Expression 3.13 for the potential energy is now a quadratic algebraic form with variables D_I. The condition that $\mathcal{P}$ be a minimum is

$$\frac{\partial \mathcal{P}}{\partial D_I} = 0. \tag{3.17}$$

Applying this condition yields a set of N linear algebraic equations in N unknowns:

$$\sum_{J=1}^{N} K_{IJ} D_J = F_I. \tag{3.18}$$

These equations must be solved numerically for the unknown nodal displacements D_J. The displacement, strain, and stress fields for an element can then be calculated from 3.1 to 3.5.

Since we have supposed the entire exterior surface to have specified loads and no specified displacements, the body can experience rigid translations and rotations. The solution to 3.18 for the unsupported structure is therefore not unique (in the absence of inertial forces). In fact, the determinant of $\mathbf{K}$ will be zero.

If, instead, the displacement is given over a part S_u of the boundary, the functions u_i must satisfy those displacement boundary conditions. Therefore, the nodal displacements D_I must have values that make u_i satisfy the conditions on S_u, at least approximately. If a node falls on a portion of the boundary, the value of a component of nodal displacement is known, and the corresponding component of nodal force is an unknown reaction. We apply a displacement boundary condition by substituting the given value of D_I in 3.18 and use the corresponding Ith equation to evaluate the reaction F_I. Therefore, if there are any displacement boundary conditions, the number N of unknown degrees of freedom of the supported body will be less than three times the number of nodes and the number of equations is similarly reduced. If the specified displacements are sufficient to prevent rigid translation and rotation, the determinate of $\mathbf{K}$ will be nonzero, and a unique solution to 3.18 will exist for the unspecified nodal displacements.

3.1.3 Basic Equations in Matrix Notation

The preceding relations can be written more concisely if the various arrays are arranged into matrices. Let us put the displacement components u_i in a column matrix **u**. There will be three rows for the three-dimensional bodies and two rows for the 2D problems. Let us also put the parameters D_K in a column matrix **D**. Then Equation 3.1, which applies to each element, becomes

$$\mathbf{u} = \mathbf{N}\mathbf{D} \tag{3.19}$$

where the shape functions N_{iK} are arranged in a rectangular matrix. The number of rows in matrix **D** is equal to the number of displacement parameters for the element. If we use the scheme in 2.79 or 2.80 to arrange the components of the strain tensor in a column matrix, then Equation 3.3 becomes

$$\boldsymbol{\varepsilon} = \mathbf{A}\mathbf{D} \tag{3.20}$$

where functions A_{ijK} have been arranged in a rectangular matrix. The constitutive relations have the form 2.81, which is repeated here:

$$\boldsymbol{\tau} = \boldsymbol{l} + \mathbf{C}\boldsymbol{\varepsilon}. \tag{3.21}$$

Equation 3.5 then becomes

$$\boldsymbol{\tau} = \boldsymbol{l} + \mathbf{B}\mathbf{D} \tag{3.22}$$

where

$$\mathbf{B} = \mathbf{C}\mathbf{A}, \tag{3.23}$$

all expressed as rectangular matrices. The element stiffness matrices in 3.11 are rectangular matrices given by

$$\mathbf{k}^m = \int_{\mathcal{V}_m} \mathbf{B}^{\mathrm{T}}\mathbf{A}\,\mathrm{d}V. \tag{3.24}$$

The nodal forces contributed by the mth element (3.12) are given by

$$\mathbf{f}^m = \int_{S_m} \mathbf{N}^{\mathrm{T}}\mathbf{T}_0\,\mathrm{d}A + \int_{\mathcal{V}_m} \mathbf{N}^{\mathrm{T}}\mathbf{b}\,\mathrm{d}V - \int_{\mathcal{V}_m} \mathbf{A}^{\mathrm{T}}\boldsymbol{l}\,\mathrm{d}V. \tag{3.25}$$

The element matrices $\mathbf{k}_m$ and $\mathbf{f}_m$ are merged to obtain the global stiffness matrix **K** and global force matrix **F**, and the final equations to be solved (3.18) can be written as

$$\mathbf{KD} = \mathbf{F}. \tag{3.26}$$

Matrix **D** now includes the full list of N unknown displacement parameters for the whole body, and force matrix **F** has an equal number of rows. The global stiffness matrix **K** is a symmetric and banded square matrix with N rows and N columns, where N is the number of unknown displacements.

3.1.4 Basic Equations Using Virtual Work

Instead of using the theorem of minimum potential energy to establish the equilibrium equations, one may use the virtual work theorem that is expressed in matrix form by 2.86. Applied to the assemble of finite elements, we have

$$\sum_{m=1}^{M} \int_{\mathcal{S}_m} \bar{\mathbf{u}}^{\mathrm{T}} \mathbf{T}\, \mathrm{d}A + \sum_{m=1}^{M} \int_{\mathcal{V}_m} \bar{\mathbf{u}}^{\mathrm{T}} \mathbf{b}\, \mathrm{d}V = \sum_{m=1}^{M} \int_{\mathcal{V}_m} \bar{\boldsymbol{\varepsilon}}^{\mathrm{T}} \boldsymbol{\tau}\, \mathrm{d}V. \tag{3.27}$$

Surface $\mathcal{S}_m$ is that part of the surface of element m that is a part of the exterior surface of the body (if any).

The shape functions 3.1 used to approximate the actual displacements can also be used to generate a compatible displacement field:

$$\bar{\mathbf{u}} = \mathbf{N}\bar{\mathbf{D}}, \tag{3.28}$$

where $\bar{\mathbf{D}}$ is the column matrix of nodal displacements of the compatible system, the so-called *virtual displacements*. Virtual displacements are numbered in the same manner as the actual nodal displacement. The compatible strains are then determined by the strain–displacement equations as in 3.3:

$$\bar{\boldsymbol{\varepsilon}} = \mathbf{A}\bar{\mathbf{D}}. \tag{3.29}$$

That is, the same coefficient matrix **A** occurs. Substituting 3.28 and 3.29 into 3.27, and merging the element matrices, we have

$$\bar{\mathbf{D}}^{\mathrm{T}} \left(\sum_{m=1}^{M} \int_{\mathcal{S}_m} \mathbf{N}^{\mathrm{T}} \mathbf{T}\, \mathrm{d}A + \sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{N}^{\mathrm{T}} \mathbf{b}\, \mathrm{d}V \right) = \bar{\mathbf{D}}^{\mathrm{T}} \left(\sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{A}^{\mathrm{T}} \boldsymbol{\tau}\, \mathrm{d}V \right). \tag{3.30}$$

Since this relation must hold for every compatible system, it must hold for arbitrary $\bar{\mathbf{D}}$. Therefore, a necessary condition for equilibrium of the finite element approximation of the stress state is

$$\sum_{m=1}^{M} \int_{\mathcal{S}_m} \mathbf{N}^{\mathrm{T}} \mathbf{T}\, \mathrm{d}A + \sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{N}^{\mathrm{T}} \mathbf{b}\, \mathrm{d}V = \sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{A}^{\mathrm{T}} \boldsymbol{\tau}\, \mathrm{d}V. \tag{3.31}$$

Note that this relation is true for any material body, whether elastic or not. It simply expresses the balance of the external applied nodal forces and the internal resisting nodal forces, **F** = **f**, where the external applied nodal forces are

$$\mathbf{F} = \sum_{m=1}^{M} \int_{S_m} \mathbf{N}^{\mathrm{T}} \mathbf{T} \, \mathrm{d}A + \sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{N}^{\mathrm{T}} \mathbf{b} \, \mathrm{d}V, \tag{3.32}$$

and the internal resisting nodal forces are

$$\mathbf{f} = \sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{A}^{\mathrm{T}} \boldsymbol{\tau} \, \mathrm{d}V. \tag{3.33}$$

This is the virtual work formula for the finite element approximation. Note that **f** here is not the same as $\mathbf{f}^m$ in Equation 3.25. The summation is in the sense of merge of the element matrices. Substituting formula 3.22 for the stress in each element, we find

$$\mathbf{f} = \left(\sum_{m=1}^{M} \int_{\mathcal{V}_m} \mathbf{A}^{\mathrm{T}} \mathbf{B} \, \mathrm{d}V \right) \mathbf{D} = \mathbf{K}\mathbf{D} \tag{3.34}$$

for an elastic material. Therefore,

$$\mathbf{F} = \mathbf{K}\mathbf{D}, \tag{3.35}$$

where **F** is, again, given by merging the element force matrices 3.12 and **K** is determined by merging the element stiffness matrices 3.11.

3.1.5 Underestimate of Displacements

We will now show that the displacements calculated by the finite element approximation to the potential energy are always "smaller" than the actual displacements. In matrix notation, the finite element approximation 3.13 to the potential energy becomes

$$\mathcal{P}^* = \frac{1}{2} \mathbf{D}^{\mathrm{T}} \mathbf{K} \mathbf{D} - \mathbf{D}^{\mathrm{T}} \mathbf{F}. \tag{3.36}$$

Since **KD** = **F**, it follows that, for the solution set, $\mathcal{P}^*$ is given by

$$\mathcal{P}^* = -\frac{1}{2} \mathbf{F}^{\mathrm{T}} \mathbf{D}. \tag{3.37}$$

Since the strains are derived from a continuous displacement field by the geometric relations and the stresses are derived from that strain field by the constitutive

equations, the finite element solution is an example of a compatible system provided that the displacement boundary conditions are exactly satisfied. The potential energy of such a compatible system is greater than the potential energy of the actual solution, which is a minimum.

Suppose that there is no temperature change so that $2\mathcal{A} = \boldsymbol{\tau}_{ij}\boldsymbol{\varepsilon}_{ij}$; then, using the virtual work formula 2.7, the potential energy (2.59) of the actual solution is

$$\mathcal{P}(\mathbf{u}) = \frac{1}{2}\int_{\mathcal{S}} T_i u_i \, \mathrm{d}A - \frac{1}{2}\int_{\mathcal{V}} b_i u_i \, \mathrm{d}V - \int_{\mathcal{S}_T} T_i^0 u_i \, \mathrm{d}A, \tag{3.38}$$

where $\mathcal{S} = \mathcal{S}_T + \mathcal{S}_u$. If the displacement boundary conditions are $u_i = 0$ on $\mathcal{S}_\mathrm{u}$, then the potential energy of the actual solution becomes

$$\mathcal{P}(\mathbf{u}) = -\frac{1}{2}\int_{\mathcal{S}_T} T_i^0 u_i \, \mathrm{d}A - \frac{1}{2}\int_{\mathcal{V}} b_i u_i \, \mathrm{d}V. \tag{3.39}$$

Since the potential of the true solution is a minimum,

$$\mathcal{P}^* > \mathcal{P}. \tag{3.40}$$

Therefore, from 3.37 and 3.39,

$$\mathbf{F}^\mathrm{T}\mathbf{D} < \int_{\mathcal{S}_T} T_i^\mathrm{o} u_i \, \mathrm{d}A + \int_{\mathcal{V}} b_i u_i \, \mathrm{d}V. \tag{3.41}$$

That is, the work $\mathbf{F}^\mathrm{T}\mathbf{D}$ of the nodal forces for the calculated displacements is less than the work of the given loads. A more precise result is obtained if the applied load is a single concentrated force. Suppose, for example, that the only load is a concentrated unit force in the x direction at the node n. Then, $F_I = 0$ except for the x component at node n, which is equal to 1. Let U_n denote the x displacement at node n, then 3.41 yields

$$U_n < u_1(\mathbf{X}_n), \tag{3.42}$$

where $\mathbf{X}_n$ is the position of node n, that is, the calculated nodal displacement is less than the actual displacement at that point.

3.1.6 Nondimensional Equations

In the numerical examples, we will usually solve the equations using any convenient values for the dimensions, material properties, and loads. The solution for any other units can be deduced from the following nondimensional equations of elasticity.

Let p have the units of stress and a have the units of length. Define nondimensional coordinates, displacements, stress, and strain by

$$\hat{\mathbf{x}} = \frac{\mathbf{x}}{a}, \quad \hat{\mathbf{u}} = \frac{E_0}{pa}\mathbf{u}, \quad \hat{\boldsymbol{\tau}} = \frac{\boldsymbol{\tau}}{p}, \quad \hat{\boldsymbol{\varepsilon}} = \frac{E_0}{p}\boldsymbol{\varepsilon}, \quad \hat{\mathbf{T}} = \frac{\mathbf{T}}{p}, \quad \hat{t} = \sqrt{\frac{E_0}{\rho a^2}}t, \tag{3.43}$$

where a, p, and E_0 are any convenient scale factors for the length scale, load magnitude, and material modulus, respectively. The new variables are then nondimensional. The basic equations of elasticity for zero body force and neglecting inertial forces are

$$\begin{aligned}
&\frac{\partial \hat{\tau}_{ij}}{\partial \hat{x}_j} = \frac{\partial^2 \hat{u}_i}{\partial \hat{t}^2}, \\
&\hat{\varepsilon}_{ij} = \frac{1}{2}\left(\frac{\partial \hat{u}_i}{\partial \hat{x}_j} + \frac{\partial \hat{u}_j}{\partial \hat{x}_i}\right), \\
&\hat{\tau}_{ij} = \frac{E/E_0}{1+\nu}\hat{\varepsilon}_{ij} + \frac{\nu E/E_0}{(1+\nu)(1-2\nu)}\hat{\varepsilon}_{kk}\delta_{ij}, \\
&\hat{\tau}_{ij} n_j = \hat{T}_i.
\end{aligned} \tag{3.44}$$

Note that the actual value of Poison's ratio must be used since it cannot be factored out of the equations. In the finite element relations, define nondimensional quantities by

$$\hat{\mathbf{D}} = \frac{E_0}{pa}\mathbf{D}, \quad \hat{\mathbf{A}} = a\mathbf{A}, \quad \hat{\mathbf{K}} = \frac{1}{aE_0}\mathbf{K}, \quad \hat{\mathbf{C}} = \frac{1}{E_0}\mathbf{C}, \quad \hat{\mathbf{F}} = \frac{1}{pa^2}\mathbf{F}. \tag{3.45}$$

The finite element relations are then

$$\begin{aligned}
&\hat{\mathbf{u}} = \mathbf{N}\hat{\mathbf{D}}, \\
&\hat{\boldsymbol{\tau}} = \hat{\mathbf{C}}\hat{\boldsymbol{\varepsilon}}, \\
&\hat{\boldsymbol{\varepsilon}} = \hat{\mathbf{A}}\hat{\mathbf{D}}, \\
&\hat{\mathbf{K}} = \int_{\hat{V}} \hat{\mathbf{A}}^{\mathrm{T}}\hat{\mathbf{C}}\hat{\mathbf{A}}\, \mathrm{d}\hat{V}, \\
&\hat{\mathbf{K}}\hat{\mathbf{D}} = \hat{\mathbf{F}}.
\end{aligned} \tag{3.46}$$

Having redefined the variables, we can then drop the ^ in 3.46. We will use nondimensional quantities in most examples. A problem is solved for any convenient values of a, p, E_0, and ρ. The actual stress and displacement can then be calculated using 3.43.

3.1.7 Uniaxial Stress

The three-dimensional body shown in Figure 3.1 has a cross section of area A. All of the stress components are zero except $\tau_{11} = \sigma(x)$. The body force is zero except $Ab_1 = p$. The differential equations of linear elasticity 2.1, 2.3, and 2.13 reduce to

$$\frac{\mathrm{d}\sigma}{\mathrm{d}x} + \frac{p}{A} = 0, \quad \varepsilon = \frac{\sigma}{E}, \quad \frac{\mathrm{d}u}{\mathrm{d}x} = \varepsilon, \tag{3.47}$$

with boundary conditions

$$u(0) = 0, \quad \sigma(L) = 0. \tag{3.48}$$

The solution for the case that p is a constant is

$$\begin{aligned} \sigma(x) &= \frac{pL}{A}\left(1 - \frac{x}{L}\right), \\ u(x) &= \frac{pL^2}{AE}\left(\frac{x}{L} - \frac{1}{2}\left(\frac{x}{L}\right)^2\right). \end{aligned} \tag{3.49}$$

The potential energy (2.59) reduces to

$$\mathcal{P}(u) = \int_0^L \frac{1}{2} AE\varepsilon^2 \,\mathrm{d}x - \int_0^L pu \,\mathrm{d}x. \tag{3.50}$$

A typical finite element is shown in Figure 3.2.

We choose linear shape functions:

$$\begin{aligned} u(x) &= N_a(x)U_a + N_b(x)U_b, \\ N_a(x) &= \frac{x_b - x}{l}, \\ N_b(x) &= \frac{x - x_a}{l}. \end{aligned} \tag{3.51}$$

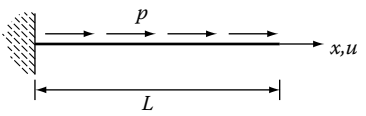


FIGURE 3.1 Uniaxial stress.

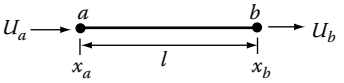


FIGURE 3.2 Axial element.

Therefore,

$$\varepsilon = \frac{1}{l}\begin{bmatrix} -1 & 1 \end{bmatrix} \mathbf{U}. \tag{3.52}$$

The potential energy (3.50) is

$$\mathcal{P}(u) = \frac{1}{2}\mathbf{U}^{\mathrm{T}}\mathbf{k}\mathbf{U} - \mathbf{U}^{\mathrm{T}}\mathbf{f} \tag{3.53}$$

where

$$\mathbf{k} = \frac{AE}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}, \ \mathbf{f} = \frac{pl}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix}. \tag{3.54}$$

For a one-element model,

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} 0 \\ U \end{bmatrix} = \frac{pL}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} + \begin{bmatrix} F_1 \\ 0 \end{bmatrix} \tag{3.55}$$

so that

$$U = \frac{pL^2}{2AE}, \ u(x) = \frac{pLx}{2AE}, \ \sigma = \frac{pL}{2A}. \tag{3.56}$$

A two-element model is shown in Figure 3.3. Merging the element stiffness and force matrices, we find the global equations:

$$\frac{AE}{l}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} 0 \\ U_2 \\ U_3 \end{bmatrix} = \frac{pl}{2}\begin{bmatrix} 1 \\ 2 \\ 1 \end{bmatrix} + \begin{bmatrix} F_1 \\ 0 \\ 0 \end{bmatrix}. \tag{3.57}$$

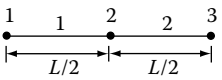


FIGURE 3.3 Two-element model.

Therefore,

$$U_2 = \frac{3}{8}\frac{pL^2}{AE},\ U_3 = \frac{1}{2}\frac{pL^2}{AE}. \tag{3.58}$$

For element 1,

$$u(x) = \frac{x}{l}U_2,\ \sigma = \frac{E}{l}U_2 = \frac{3}{2}\frac{pl}{A}. \tag{3.59}$$

For element 2,

$$u(x) = \frac{(L-x)}{l}U_2 + \frac{(x-l)}{l}U_3,\quad \sigma = \frac{E}{l}\left(U_3 - U_2\right) = \frac{pL}{4A}. \tag{3.60}$$

The results are compared with the exact solution in Figures 3.4 and 3.5 for non-dimensional variables $\frac{2AE}{pL^2}u \to u,\ \ \frac{A}{pL}\sigma \to S$. The displacement is uniformly too small. The stress is too large at some points and too small at others. The energy measure 3.41 for the two-element model is as follows:

$$\mathbf{F}^{\mathrm{T}}\mathbf{D} = \frac{pL}{4}\begin{bmatrix}1 & 2 & 1\end{bmatrix}\begin{bmatrix}0\\ U_2\\ U_3\end{bmatrix} = \frac{5}{16}\frac{p^2L^3}{AE}. \tag{3.61}$$

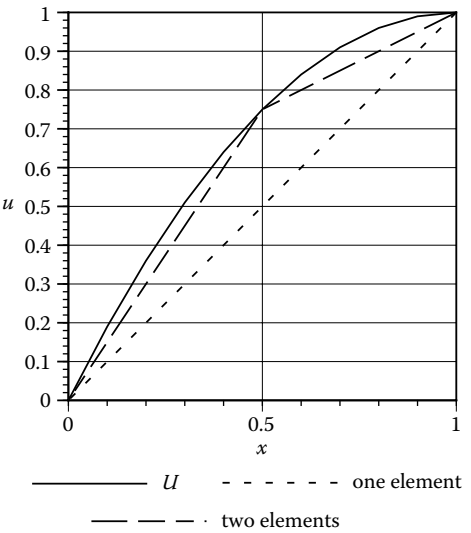


FIGURE 3.4 Displacement.

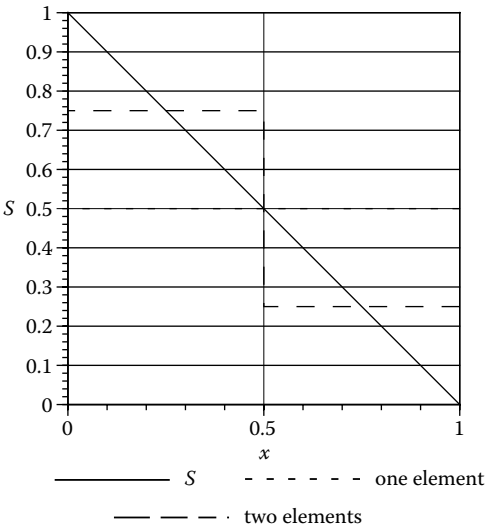


FIGURE 3.5 Stress.

Using 3.49,

$$\int_A b_i u_i \, \mathrm{d}A = \int_0^L pu \, \mathrm{d}x = \frac{1}{3} \frac{p^2 L^3}{AE}. \tag{3.62}$$

That is,

$$\mathbf{F}^{\mathrm{T}}\mathbf{D} < \int_A b_i u_i \, \mathrm{d}A \tag{3.63}$$

as required by 3.41.

Although minimum energy only ensures that the weighted average displacement as measured by 3.63 will be too small, we see from Figure 3.4 that the displacements are everywhere less than or equal to the exact solution. As indicated in Figure 3.5, the stress components at any point may be less than or greater than the exact solution. This is a typical situation.

3.2 GENERAL EQUATIONS FOR AN ASSEMBLY OF ELEMENTS

Let us consider a material body that is divided into a number of finite elements that are numbered consecutively. The symmetric stress tensor, symmetric strain tensor, and the displacement vector within element n are denoted by τ_{ij}^n, ε_{ij}^n, and u_i^n. Within that element, these functions as well as the partial derivatives of the displacement are continuous and satisfy the equations of equilibrium, the geometric relations, and the constitutive relations:

$$\tau_{ij}^{n},_{j} + b_i^n = 0 \ \text{ in } \mathcal{V}^n. \tag{3.64}$$

$$\varepsilon_{ij}^n = u_{ij}^n \ \text{ in } \mathcal{V}^n. \tag{3.65}$$

$$\tau_{ij}^n = \frac{\partial \mathcal{A}(\boldsymbol{\varepsilon})}{\partial \varepsilon_{ij}^n}, \varepsilon_{ij}^n = \frac{\partial \mathcal{B}(\boldsymbol{\tau})}{\partial \tau_{ij}^n}, \mathcal{A} + \mathcal{B} = \tau_{ij}\varepsilon_{ij} \text{ , in } \mathcal{V}^n. \tag{3.66}$$

A comma denotes the partial derivative with respect to the indicated coordinate and let $u_{ij} \equiv (u_{i,j} + u_{j,i})/2$.

If a part $\mathcal{S}_T^n$ of the boundary $\partial\mathcal{V}^n$ is an exterior boundary on which tractions are specified, then τ_{ij}^n must satisfy the following boundary conditions:

$$\tau_{ij}^n n_j^n = T_i^0 \text{ on } \mathcal{S}_T^n, \tag{3.67}$$

If a part $\mathcal{S}_u^n$ of $\partial\mathcal{V}^n$ is an exterior boundary on which displacements are specified, then u_i^n must satisfy the following boundary conditions:

$$u_i^n = u_i^0 \text{ on } \mathcal{S}_u^n. \tag{3.68}$$

The remaining (interior) part of the boundary of element n is $\mathcal{S}^n$. We assume here for simplicity that the parts of the surface are distinct: $\partial\mathcal{V}^n = \mathcal{S}_T^n + \mathcal{S}_u^n + \mathcal{S}^n$. The traction vector on the boundary of element n is

$$\tau_{ij}^n n_j^n = T_i^n \text{ on } \mathcal{S}^n \text{ and } \mathcal{S}_u^n. \tag{3.69}$$

Two contiguous elements share the interior boundary $\mathcal{S}^n$ and they must have a common displacement U_i on that surface:

$$u_i^n = U_i \text{ on } \mathcal{S}^n. \tag{3.70}$$

Also, the stress vectors from the two adjacent elements must balance any tractions applied to that surface and we will assume for simplicity that there are none:

$$T_i^n + T_i^{n+} = 0 \text{ on } \mathcal{S}^n. \tag{3.71}$$

We will need to use integration by parts within an element where the functions are required to be continuous:

$$\begin{aligned} \int_{\mathcal{V}^n} \tau_{ij}^n \overline{u}_{ij}^n \, \mathrm{d}V &= \int_{\mathcal{V}^n} \left\{ \left(\tau_{ij}^n \overline{u}_i^n \right),_j - \tau_{ij}^n,_j \, \overline{u}_i^n \right\} \mathrm{d}V \\ &= \int_{\partial\mathcal{V}^n} \overline{u}_i^n \tau_{ij}^n n_j^n \, \mathrm{d}A - \int_{\mathcal{V}^n} \tau_{ij}^n,_j \, \overline{u}_i^n \, \mathrm{d}V \end{aligned} \tag{3.72}$$

for any smooth function $\overline{u}_i^n$.

3.2.1 Generalized Variational Principle

All differential equations and boundary conditions can be replaced by a single variational principle. Let us define a function $\mathcal{W}$ of the fields $u_i^n, \varepsilon_{ij}^n, \tau_{ij}^n, U_i$, *and* T_i^n:

$$\mathcal{W} = \sum_{n} \left\{ \int_{\mathcal{V}^n} \left[\mathcal{A}^n(\boldsymbol{\varepsilon}^n) - \varepsilon_{ij}^n \tau_{ij}^n + \tau_{ij}^n u_{ij}^n - b_i^n u_i^n \right] \mathrm{d}V \right.$$
$$\left. - \int_{\mathcal{S}^n} T_i^n \left(u_i^n - U_i \right) \mathrm{d}A - \int_{\mathcal{S}_u^n} T_i^n \left(u_i^n - u_i^0 \right) \mathrm{d}A - \int_{\mathcal{S}_T^n} T_i^0 u_i^n \, \mathrm{d}A \right\}. \tag{3.73}$$

Note that u_i^n, ε_{ij}^n, and τ_{ij}^n are not required to be continuous between elements. Also, note that each surface $\mathcal{S}^n$ is shared by two elements, so the integral over each $\mathcal{S}^n$ occurs twice in the summation. The functional $\mathcal{W}$ is known as the generalized Hu–Washizu functional. We now show that the functions making the first derivative of $\mathcal{W}$ zero will satisfy the basic equations of elasticity 3.64 to 3.71. The condition that the first derivative (first variation) of $\mathcal{W}$ is zero is therefore equivalent to the differential equations, boundary conditions, and continuity conditions of linear elasticity.

To calculate the derivative of $\mathcal{W}$, we replace $u_i^n, \varepsilon_{ij}^n, \tau_{ij}^n, U_i$, and T_i^n by $u_i^n + \alpha \bar{u}_i^n$, $\varepsilon_{ij}^n + \alpha \bar{\varepsilon}_{ij}^n$, $\tau_{ij}^n + \alpha \bar{\tau}_{ij}^n$, $U_i + \alpha \bar{U}_i$, and $T_i^n + \alpha \bar{T}_i^n$, and then calculate the ordinary derivative with respect to the real variable α, evaluated at $\alpha = 0$. We find

$$D\mathcal{W} = \sum_{n} \left\{ \int_{\mathcal{V}^n} \left[\left(\frac{\partial \mathcal{A}(e)}{\partial \varepsilon_{ij}^n} - \tau_{ij}^n \right) \bar{\varepsilon}_{ij}^n + \left(u_{ij}^n - \varepsilon_{ij}^n \right) \bar{\tau}_{ij}^n - \left(\tau_{ij,j}^n + b_i^n \right) \bar{u}_i^n \right] \mathrm{d}V \right.$$
$$- \int_{\mathcal{S}^n} \left[(u_i^n - U_i) \bar{T}_i^n + (T_i^n - \tau_{ij}^n n_j^n) \bar{u}_i^n - T_i^n \bar{U}_i \right] \mathrm{d}A$$
$$- \int_{\mathcal{S}_u^n} \left[\left(u_i^n - u_i^0 \right) \bar{T}_i^n + \left(T_i^n - \tau_{ij}^n n_j^n \right) \bar{u}_i^n \right] \mathrm{d}A \tag{3.74}$$
$$\left. - \int_{\mathcal{S}_T^n} \left(T_i^0 - \tau_{ij}^n n_j^n \right) \bar{u}_i^n \, \mathrm{d}A \right\}.$$

We now require that $D\mathcal{W} = 0$ for all functions $\bar{u}_i^n, \bar{\varepsilon}_{ij}^n, \bar{\tau}_{ij}^n, \bar{U}_i$, and $\bar{T}_i^n$. We obtain 3.64 to 3.70 immediately. Furthermore, for adjacent elements, the integrals over their common boundary segment can be combined to obtain

$$\int_{\mathcal{S}^n} T_i^n \bar{U}_i \, \mathrm{d}A + \int_{\mathcal{S}^{n+}} T_i^{n+} \bar{U}_i \, \mathrm{d}A = \int_{\mathcal{S}^n} (T_i^n + T_i^{n+}) \bar{U}_i \, \mathrm{d}A = 0, \tag{3.75}$$

from which 3.71 follows. Thus, all equations of elasticity are conditions for the vanishing of the first derivative of $\mathcal{W}$, and instead of solving those equations directly, we could seek functions such that $D\mathcal{W} = 0$. We next consider the special cases when the functions $u_i^n, \varepsilon_{ij}^n, \tau_{ij}^n, U_i$, and T_i^n are related by one or more of Equations 3.64 to 3.66.

3.2.2 Potential Energy

Let the strain tensor be defined by the geometric relations 3.65, let the displacements satisfy the displacement boundary conditions 3.68, and let the displacements be continuous between elements so that 3.70 is satisfied. Then, the general functional 3.73 reduces to a function of the displacements that is called the potential energy:

$$\mathcal{P}(\mathbf{u}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[\mathcal{A}^n(\boldsymbol{\varepsilon}^n) - b_i^n u_i^n \right] \mathrm{d}V - \int_{S_T^n} T_i^0 u_i^n \mathrm{d}A \right\}. \tag{3.76}$$

The stress tensor no longer occurs in the functional. It is defined by the constitutive relations 3.66 and the stress vector is defined by 3.69. The derivative of the potential energy is

$$D\mathcal{P} = \sum_n \left\{ \int_{\mathcal{V}^n} \left[-(\tau_{ij}^n{}_{,j} + b_i^n) \bar{u}_i^n \right] \mathrm{d}V + \int_{\mathcal{S}^n} \left[\tau_{ij}^n n_j^n \bar{u}_i^n \right] \mathrm{d}A - \int_{S_T^n} (T_i^0 - \tau_{ij}^n n_j^n) \bar{u}_i^n \, \mathrm{d}A \right\}. \tag{3.77}$$

The condition that $D\mathcal{P}$ is zero for all functions $\bar{\mathbf{u}}$ satisfying the displacement boundary conditions implies the equilibrium equations 3.64 and 3.71, and the stress boundary conditions 3.67. This functional is the basis for the displacement method of finite element analysis, which we have heretofore followed.

3.2.3 Hybrid Displacement Functional

Let the strain tensor be defined by geometric relations 3.65 and let the displacements satisfy displacement boundary conditions 3.68, but retain the possibility that the displacements are not continuous between elements. In this case, the general functional 3.73 reduces to a functional of the displacements $\mathbf{u}$ in $\mathcal{V}^n$ and the displacements $\mathbf{U}$ and stress vector $\mathbf{T}$ on the internal element boundaries $\mathcal{S}^n$:

$$\mathcal{W}_{\mathrm{HD}}(\mathbf{u},\mathbf{U},\mathbf{T}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[\mathcal{A}^n(\boldsymbol{\varepsilon}^n) - b_i^n u_i^n \right] \mathrm{d}V - \int_{\mathcal{S}^n} T_i^n \left(u_i^n - U_i \right) \mathrm{d}A - \int_{\mathcal{S}_T^n} T_i^0 u_i^n \, \mathrm{d}A \right\}. \tag{3.78}$$

The stress tensor does not occur in the functional. It is defined by the constitutive relations 3.66. The derivative of the functional is

$$\begin{aligned} D\mathcal{W}_{\mathrm{HD}} = \sum_n \Bigg\{ & \int_{\mathcal{V}^n} \left[-(\tau_{ij,j}^n + b_i^n)\bar{u}_i^n \right] \mathrm{d}V \\ & - \int_{\mathcal{S}^n} \left[(u_i^n - U_i)\bar{T}_i^n + (T_i^n - \tau_{ij}^n n_j^n)\bar{u}_i^n - T_i^n \bar{U}_i \right] \mathrm{d}A \\ & - \int_{\mathcal{S}_u^n} \tau_{ij}^n n_j^n \bar{u}_i^n \, \mathrm{d}A - \int_{\mathcal{S}_T^n} (T_i^0 - \tau_{ij}^n n_j^n)\bar{u}_i^n \, \mathrm{d}A \Bigg\}. \end{aligned} \tag{3.79}$$

The condition that $D\mathcal{W}_{\mathrm{HD}}$ is zero for all functions $\bar{\mathbf{u}}$ satisfying the displacement boundary conditions, all $\mathbf{U}$, and all $\mathbf{T}$ implies Equations 3.64, 3.67, 3.69, 3.70, and 3.71. This functional is the basis for the so-called hybrid displacement methods. It can be used for a rational formulation of the finite element method using nonconforming elements.

3.2.4 Hybrid Stress and Complementary Energy

Let the strain tensor be determined by constitutive relations 3.66. Let the stress vector be defined by 3.69 and require the stress tensor to satisfy the equilibrium equations 3.64. The general functional reduces to

$$\begin{aligned} \mathcal{W}_{\mathrm{HS}} = \sum_n \Bigg\{ & \int_{\mathcal{V}^n} \left[-\mathcal{B}^n(\boldsymbol{\tau}^n) + \tau_{ij}^n u_{ij}^n - b_i^n u_i^n \right] \mathrm{d}V - \int_{\mathcal{S}^n} T_i^n \left(u_i^n - U_i \right) \mathrm{d}A \\ & - \int_{\mathcal{S}_u^n} T_i^n \left(u_i^n - u_i^0 \right) \mathrm{d}A - \int_{\mathcal{S}_T^n} T_i^0 u_i^n \, \mathrm{d}A \Bigg\}. \end{aligned} \tag{3.80}$$

Applying formula 3.72 for integrating by parts, we obtain

$$\mathcal{W}_{\mathrm{HS}}(\mathbf{U},\boldsymbol{\tau}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[-\mathcal{B}^n(\boldsymbol{\tau}^n) \right] \mathrm{d}V + \int_{\mathcal{S}^n} T_i^n U_i \, \mathrm{d}A + \int_{\mathcal{S}_u^n} T_i^n u_i^0 \, \mathrm{d}A + \int_{\mathcal{S}_T^n} (T_i - T_i^0) U_i^n \, \mathrm{d}A \right\}. \tag{3.81}$$

Since **u** occurs only on the boundary, we have replaced it by **U**. The functional is defined only for those stress systems that satisfy the equilibrium equations. Before calculating the derivative, we can extend the definition to include all smooth functions τ_{ij} via Lagrangian multipliers u_i applied to the subsidiary conditions 3.64:

$$\bar{\mathcal{W}}_{\mathrm{HS}}(\mathbf{U},\boldsymbol{\tau}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[-\mathcal{B}^n(\boldsymbol{\tau}^n) - (\tau_{ij}^n{}_{,j} + b_i^n)u_i \right] \mathrm{d}V + \int_{S^n} T_i^n U_i \, \mathrm{d}A \right.$$
$$\left. + \int_{S_u^n} T_i^n u_i^0 \, \mathrm{d}A + \int_{S_T^n} (T_i - T_i^0) U_i^n \, \mathrm{d}A \right\}. \tag{3.82}$$

The derivative of this functional is

$$D\bar{\mathcal{W}}_{\mathrm{HS}} = \sum_n \left\{ \int_{\mathcal{V}^n} \left[(u_{ij}^n - \varepsilon_{ij}^n)\bar{\tau}_{ij}^n - (\tau_{ij}^n{}_{,j} + b_i^n)\bar{u}_i^n \right] \mathrm{d}V \right.$$
$$- \int_{S^n} \left[(u_i^n - U_i)\bar{T}_i^n - T_i^n \bar{U}_i \right] \mathrm{d}A$$
$$- \int_{S_u^n} \left[(u_i^n - u_i^0)\bar{T}_i^n \right] \mathrm{d}A \tag{3.83}$$
$$\left. - \int_{S_T^n} \left[(T_i^0 - T_i)\bar{U}_i + (u_i^n - U_i)\bar{T}_i^n \right] \mathrm{d}A \right\}.$$

As the notation suggests, the Lagrangian multipliers are the displacement components and vanishing of the derivative is equivalent to Equations 3.65, 3.67, 3.68, 3.70, and 3.71. The functional $\bar{\mathcal{W}}_{\mathrm{HS}}(\mathbf{U},\boldsymbol{\tau})$ is the basis for the hybrid stress method of finite element analysis. Shape functions must be selected for the stress tensor that satisfy the equations of equilibrium, and shape functions must be specified for the interelement displacements.

If we further restrict the stress tensor to satisfy the stress boundary conditions 3.67 and the interelement equilibrium conditions 3.71, the function reduces to the complementary energy:

$$C(\boldsymbol{\tau}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[\mathcal{B}^n(\boldsymbol{\tau}^n) \right] \mathrm{d}V - \int_{S_u^n} T_i^n u_i^0 \, \mathrm{d}A \right\}. \tag{3.84}$$

In this case, the stress tensor and the stress vector must satisfy conditions 3.64, 3.67, 3.69, and 3.71. The strain tensor is defined by 3.66. Vanishing of the derivative of **C** ensures the existence of displacements satisfying 3.65, 3.68, and 3.70, that is, the compatibility conditions and support conditions are satisfied.

3.2.5 Mixed Methods of Analysis

Let us now modify the general functional by using the constitutive relations 3.66 to eliminate the strain tensor and also define the stress vector by 3.69. The result is called the generalized Hellinger–Reissner function:

$$\bar{\mathcal{R}}(\mathbf{u},\mathbf{U},\boldsymbol{\tau}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[-\mathcal{B}(\boldsymbol{\tau}) + \tau_{ij}^n u_{ij}^n - b_i^n u_i^n \right] \mathrm{d}V \right.$$
$$\left. - \int_{S^n} T_i^n \left(u_i^n - U_i \right) \mathrm{d}A - \int_{S_u^n} T_i^n \left(u_i^n - u_i^0 \right) \mathrm{d}A - \int_{S_T^n} T_i^0 u_i^n \, \mathrm{d}A \right\}. \tag{3.85}$$

The derivative of $\bar{\mathcal{R}}$ is

$$D\bar{\mathcal{R}} = \sum_n \left\{ \int_{\mathcal{V}^n} \left[(u_{ij}^n - \varepsilon_{ij}^n)\bar{\tau}_{ij}^n - (\tau_{ij}^n{}_{,j} + b_i^n)\bar{u}_i^n \right] \mathrm{d}V - \int_{S^n} \left[(u_i^n - U_i)\bar{T}_i^n - T_i^n \bar{U}_i \right] \mathrm{d}A \right.$$
$$\left. - \int_{S_u^n} (u_i^n - u_i^0)\bar{T}_i^n \, \mathrm{d}A - \int_{S_T^n} (T_i^0 - T_i)\bar{u}_i^n \, \mathrm{d}A \right\}. \tag{3.86}$$

Vanishing of the derivative is equivalent to the remaining equations, 3.64, 3.65, 3.67, 3.68, 3.70, and 3.71. The functional $\bar{\mathcal{R}}$ depends on the functions $\boldsymbol{\tau}$ and **u** defined within each element and the functions **U** defined on the boundaries between elements. The displacement field need not be continuous, nor does the stress field need to satisfy the equilibrium equations. If we do require continuous displacement fields so that 3.70 is satisfied, the functional reduces to the Reissner function:

$$\mathcal{R}(\mathbf{u},\boldsymbol{\tau}) = \sum_n \left\{ \int_{\mathcal{V}^n} \left[\tau_{ij}^n \varepsilon_{ij}^n - \mathcal{B}(\boldsymbol{\tau}^n) - b_i^n u_i^n \right] \mathrm{d}V - \int_{S_T^n} T_i^0 u_i^n \, \mathrm{d}A \right\}. \tag{3.87}$$

This function is the basis for the so-called mixed methods of finite element analysis in which shape functions are specified for both stress and displacement.

In the displacement method, we described the displacement within each element by shape functions and used the potential energy function. In the mixed method, we describe both the displacement and the stress in the elements by shape functions, and use the Reissner function and choose continuous displacements that satisfy the displacement boundary conditions.

Let the displacement in an element be described by shape functions **N** and displacement parameters **D**:

$$u_i^n = N_{iK}^n D_K^n,\ u_{ij}^n = A_{ijK}^n D_K^n . \tag{3.88}$$

In this section, the summation convention applies to repeated subscripts and summation is over the appropriate range, a range that is different for each alphabet. The shape functions **N** must be such that the displacements are continuous between elements for all values of the parameters **D**. This can be accomplished by using the same shape functions as for the displacement method. Second, let the stress in an element be described by shape functions **H** and stress parameters **S**:

$$\tau_{ij}^n = H_{ij\alpha}^n S_\alpha^n . \tag{3.89}$$

The complementary strain energy density $\mathcal{B}$ is then

$$\mathcal{B} = \frac{1}{2}\tau_{ij}^n c_{ijkm}^n \tau_{km}^n = \frac{1}{2} c_{ijkm}^n H_{ij\alpha}^n S_\alpha^n H_{km\beta}^n S_\beta^n . \tag{3.90}$$

Substituting into the Reissner functional gives

$$\mathcal{R} = \sum_n \left\{ S_\alpha^n Q_{\alpha K}^n D_K^n - \frac{1}{2} S_\alpha^n f_{\alpha\beta}^n S_\beta^n - F_K^n D_K^n \right\}, \tag{3.91}$$

where

$$B_{K\alpha}^n = \int_{\mathcal{V}^n} H_{ij\alpha}^n A_{ijK}^n \, \mathrm{d}V, \tag{3.92}$$

$$f_{\alpha\beta}^n = \int_{\mathcal{V}^n} c_{ijkm}^n H_{ij\alpha}^n H_{km\beta}^n \, \mathrm{d}V . \tag{3.93}$$

$$F_K^n = \int_{S_T^n} T_i^0 N_{iK}^n \, \mathrm{d}A + \int_{\mathcal{V}^n} b_i N_{iK}^n \, \mathrm{d}V . \tag{3.94}$$

Merging the element matrices for all elements that share displacement parameters and/or stress parameters, we find in matrix notation that

$$\mathcal{R}(\mathbf{S},\mathbf{D}) = \mathbf{S}^{\mathrm{T}}\mathbf{B}^{\mathrm{T}}\mathbf{D} - \frac{1}{2}\mathbf{S}^{\mathrm{T}}\mathbf{f}\mathbf{S} - \mathbf{F}^{\mathrm{T}}\mathbf{S}. \tag{3.95}$$

The condition that the first derivative of $\mathcal{R}$ be zero implies that the partial derivative with respect to each of the algebraic parameters is zero. This provides two sets of equations:

$$\mathbf{B}\mathbf{S} = \mathbf{F}, \tag{3.96}$$

and

$$\mathbf{B}^{\mathrm{T}}\mathbf{D} = \mathbf{f}\mathbf{S}. \tag{3.97}$$

The first is the equilibrium equation. The left-hand side of the second is a formula for the generalized strains as derived from the geometry of deformation, and the right-hand side is a formula for the generalized strains as derived from the constitutive relations in terms of the symmetric flexibility matrix **f**. This leads to the redundant force method described in Section 1.6 (Chapter 1).

There will be s stress parameters and d displacement parameters. The matrix **B** is $d \times s$. If $s = d$, the equilibrium equations can be solved for **S** unless the *det* **B** = 0. If the determinant is zero, the system is typically kinematically unstable. If $d > s$, then **S** is overdetermined and usually no solution exists to the equilibrium equations. This implies the existence of displacement fields with zero generalized strains, so-called spurious kinematical modes. In general then, $d < s$ and the equilibrium equations 3.96 have more unknowns S_α than equations. The general solution to 3.96 is then of the form

$$\mathbf{S} = \mathbf{b}_0\mathbf{F} + \mathbf{b}_1\mathbf{X}. \tag{3.98}$$

The parameters **X** are $s - d = r$ in number and are called *redundant forces*. The coefficient matrices are such that

$$\mathbf{B}\mathbf{b}_0 = \mathbf{1} \tag{3.99}$$

and

$$\mathbf{B}\mathbf{b}_1 = \mathbf{0}. \tag{3.100}$$

Multiplying 3.97 by $\mathbf{b}_1^{\mathrm{T}}$ provides the compatibility condition

$$\mathbf{b}_1^{\mathrm{T}}\mathbf{f}\mathbf{S} = \mathbf{0}. \tag{3.101}$$

Substituting 3.98 into 3.101 yields the basic equations of the redundant force method:

$$\mathbf{S} = \mathbf{bF}, \tag{3.102}$$

where

$$\begin{aligned} \mathbf{b} &= \mathbf{b}_0 - \mathbf{b}_1\mathbf{c}_1^{-1}\mathbf{c}_0 \\ \mathbf{c}_0 &= \mathbf{b}_1^{\mathrm{T}}\mathbf{f}\mathbf{b}_0, \\ \mathbf{c}_1 &= \mathbf{b}_1^{\mathrm{T}}\mathbf{f}\mathbf{b}_1. \end{aligned} \tag{3.103}$$

There exists standard algorithms for construction of solution 3.98 of the equilibrium equations, and the mixed method is a perfectly workable procedure for stress analysis provided someone has developed a computer code to automate the calculations. The truss example in Section 1.6 is an example of the redundant force calculations.

Alternatively, the stiffness formulation can be recovered from Equations 3.96 and 3.97:

$$\mathbf{KD} = \mathbf{F} \tag{3.104}$$

where

$$\mathbf{K} = \mathbf{B}\mathbf{f}^{-1}\mathbf{B}^{\mathrm{T}}. \tag{3.105}$$

Since the displacements allowed in the Reissner functional are required to satisfy the displacement boundary conditions, this **K** is the reduced stiffness matrix obtained after applying the support conditions.

3.3 NEARLY INCOMPRESSIBLE MATERIALS

As the Poisson ratio approaches ½, the material constant λ and the bulk modulus become infinite, that is, the volumetric strain approaches zero. The solution to the elasticity equations becomes strongly dependent on the Poisson ratio and the finite element equations may be poorly conditioned for numerical solution. In such cases, a mixed method, which we will now develop, may be necessary.

We first recast the elasticity equations so that they hold in the limit as the Poisson ratio approaches ½. The deviatoric stress, deviatoric strain, volumetric strain, and mean pressure are defined as follows.

$$\begin{aligned} e_{ij} &= \varepsilon_{ij} - \frac{1}{3}\varepsilon\delta_{ij}, \quad \varepsilon = \varepsilon_{kk}, \\ s_{ij} &= \tau_{ij} + p\delta_{ij}, \quad p = -\frac{1}{3}\tau_{kk}. \end{aligned} \tag{3.106}$$

The constitutive relations for an isotropic material can be expressed as

$$\begin{aligned} s_{ij} &= 2\mu e_{ij}, \\ \varepsilon &= -\frac{p}{\kappa}, \end{aligned} \tag{3.107}$$

where κ is the bulk modulus. These relations are then well behaved as the Poisson ratio approaches ½ and $\kappa \to \infty$.

With displacements and mean pressure as the fundamental unknowns, the equations of elasticity that have to be solved are

$$\begin{aligned} 2\mu \frac{\partial e_{ij}}{\partial x_j} - \frac{\partial p}{\partial x_i} + b_i &= 0, \\ \varepsilon + \frac{p}{\kappa} &= 0, \end{aligned} \tag{3.108}$$

where

$$\begin{aligned} \varepsilon &= \frac{\partial u_k}{\partial x_k}, \\ e_{ij} &= \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}. \end{aligned} \tag{3.109}$$

The corresponding virtual work formula can be discovered by multiplying Equations 3.108 by virtual displacements $\overline{u}_i$ and virtual pressure $\overline{p}$ and integrating over the volume. After integration by parts, we find

$$\int_V \left\{ \left(2\mu e_{ij} - p\delta_{ij}\right)\overline{\varepsilon}_{ij} - \left(\varepsilon + \frac{p}{\kappa}\right)\overline{p} \right\} \mathrm{d}V = \int_A T_i \overline{u}_i \, \mathrm{d}A + \int_V b_i \overline{u}_i \, \mathrm{d}V, \tag{3.110}$$

where

$$T_i = n_j \tau_{ij} \tag{3.111}$$

is the surface traction vector. Substituting

$$\overline{e}_{ij} = \overline{\varepsilon}_{ij} - \frac{1}{3}\overline{\varepsilon}\delta_{ij} \tag{3.112}$$

into 3.110, we find

$$\int_V \left\{ 2\mu e_{ij}\overline{e}_{ij} - p\overline{\varepsilon} - \left(\varepsilon + \frac{p}{\kappa}\right)\overline{p} \right\} \mathrm{d}V = \int_A T_i \overline{u}_i \, \mathrm{d}A + \int_V b_i \overline{\boldsymbol{u}}_i \, \mathrm{d}V. \tag{3.113}$$

Equation 3.113 is equivalent to the stationary value of the functional

$$\mathcal{M}(\mathbf{u}, p) = \int_{\mathcal{V}^n} \left[\frac{1}{2} \left(2\mu e_{ij} e_{ij} - \frac{1}{\kappa} (p)^2 \right) - \varepsilon p - b_i u_i \right] \mathrm{d}V - \int_{\mathcal{S}_T^n} T_i^0 u_i \, \mathrm{d}A \tag{3.114}$$

defined for those displacements that satisfy the displacement boundary conditions. The conditions for a zero first derivative are the equilibrium equations, the boundary conditions on stress, and the constitutive equation for volume change. Formula 3.114 could also be derived from the general mixed formula 3.87 by using 3.106.

The finite element formulations for the displacements are as set forth in 3.1:

$$u_i(\mathbf{x}) = \sum_{K \in \mathbf{I}_m} N_{iK}(\mathbf{x}) D_K, \quad \bar{u}_i(\mathbf{x}) = \sum_{K \in \mathbf{I}_m} N_{iK}(\mathbf{x}) \bar{D}_K. \tag{3.115}$$

From 3.3,

$$\varepsilon_{ij} = \sum_{K \in \mathbf{I}_m} A_{ijK} D_K, \quad \bar{\varepsilon}_{ij} = \sum_{K \in \mathbf{I}_m} A_{ijK} \bar{D}_K. \tag{3.116}$$

Therefore,

$$\begin{aligned} \varepsilon &= \sum_{K \in \mathbf{I}_m} A_{iiK} D_K, \\ \bar{\varepsilon} &= \sum_{K \in \mathbf{I}_m} A_{iiK} \bar{D}_K, \\ e_{ij} &= \sum_{K \in \mathbf{I}_m} \left(A_{ijK} - \frac{1}{3} A_{iiK} \right) D_K = \sum_{K \in \mathbf{I}_m} A_{ijK}^{\mathrm{D}} D_K, \\ \bar{e}_{ij} &= \sum_{K \in \mathbf{I}_m} A_{ijK}^{\mathrm{D}} \bar{D}_K. \end{aligned} \tag{3.117}$$

In addition, we need finite element representations for p and $\bar{p}$. In general, we use different shape functions:

$$p = H_\alpha P_\alpha, \quad \bar{p} = H_\alpha \bar{P}_\alpha. \tag{3.118}$$

Then, substituting into 3.113 and merging the element matrices, we find

$$\begin{aligned} &\sum_K \sum_M K_{KM}^{11} D_M \bar{D}_K + \sum_K \sum_\beta K_{K\beta}^{12} P_\beta \bar{D}_K \\ &\quad + \sum_\alpha \sum_M K_{\alpha M}^{21} D_M \bar{P}_\alpha + \sum_\alpha \sum_{\boldsymbol{\beta}} K_{\alpha\beta}^{22} P_\beta \bar{P}_\alpha = \sum_K F_K \bar{D}_K, \end{aligned} \tag{3.119}$$

where

$$\begin{aligned} K_{KM}^{11} &= \int_{\mathrm{V}} 2\mu A_{ijK}^{D} A_{ijM}^{D}\, \mathrm{d}V, \\ K_{K\beta}^{12} &= -\int_{V} A_{iiK} H_{\beta}\, \mathrm{d}V, \\ K_{\alpha M}^{21} &= -\int_{V} H_{\alpha} A_{iiM}\, \mathrm{d}V, \\ K_{\alpha\beta}^{22} &= -\int_{V} H_{\alpha} H_{\beta}\, \mathrm{d}V. \end{aligned} \tag{3.120}$$

Equation 3.119 must hold identically for all virtual parameters. Therefore,

$$\mathbf{K}^{11}\mathbf{D} + \mathbf{K}^{12}\mathbf{P} = \mathbf{F}, \tag{3.121}$$

and

$$\mathbf{K}^{21}\mathbf{D} + \mathbf{K}^{22}\mathbf{P} = \mathbf{0}. \tag{3.122}$$

3.3.1 Nearly Incompressible Plane Strain

In the case of plane strain, the constitutive relations with mean stress as a separate unknown have the form

$$\begin{aligned} \tau_{\alpha\beta} &= 2\mu e_{\alpha\beta} - p\delta_{\alpha\beta}, \\ \varepsilon_{\theta\theta} &= -\frac{p}{\kappa}. \end{aligned} \tag{3.123}$$

The equations to be solved for displacements u_α and mean pressure p are

$$\begin{aligned} 2\mu \frac{\partial e_{\alpha\beta}}{\partial x_{\beta}} - \frac{\partial p}{\partial x_{\beta}} + b_{\beta} &= 0, \\ \varepsilon + \frac{p}{\kappa} &= 0, \end{aligned} \tag{3.124}$$

where

$$\begin{aligned} \varepsilon_{\theta\theta} &= \frac{\partial u_{\theta}}{\partial x_{\theta}}, \\ e_{\alpha\beta} &= \frac{1}{2}\left(\frac{\partial u_{\alpha}}{\partial x_{\beta}} + \frac{\partial u_{\beta}}{\partial x_{\alpha}} \right) - \frac{1}{3}\frac{\partial u_{\theta}}{\partial x_{\theta}}\delta_{\alpha\beta}. \end{aligned} \tag{3.125}$$

Multiply Equations 3.123 by virtual displacements $\bar{u}_\alpha$ and virtual pressure $\bar{p}$ and integrating over the area. After integration by parts, we find

$$\int_{\mathcal{A}} \left\{ 2\mu e_{\alpha\beta} \frac{\partial \bar{u}_\alpha}{\partial x_\beta} - p \frac{\partial \bar{u}_\alpha}{\partial x_\alpha} - \left(\varepsilon_{\alpha\alpha} + \frac{p}{\kappa} \right) \bar{p} \right\} \mathrm{d}A = \int_C T_\alpha \bar{u}_\alpha \, \mathrm{d}s + \int_{\mathcal{A}} b_\alpha \bar{u}_\alpha \, \mathrm{d}A. \quad (3.126)$$

Other choices of parameters are possible. For example, in place of 3.123 we can write

$$\begin{aligned} \tau_{\alpha\beta} &= 2\mu\varepsilon_{\alpha\beta} - \beta p \delta_{\alpha\beta}, \\ \varepsilon_{\theta\theta} &= -\frac{p}{\kappa}, \end{aligned} \quad (3.127)$$

where

$$\beta = \frac{\lambda}{\kappa} = \frac{3\nu}{1+\nu}. \quad (3.128)$$

The virtual work formula 3.126 can then be put in the form

$$\int_{\mathcal{A}} \left\{ 2\mu\varepsilon_{\alpha\beta} \frac{\partial \bar{u}_\alpha}{\partial x_\beta} - \beta p \frac{\partial \bar{u}_\alpha}{\partial x_\alpha} - \left(\varepsilon_{\alpha\alpha} + \frac{p}{\kappa} \right) \beta \bar{p} \right\} \mathrm{d}A = \int_C T_\alpha \bar{u}_\alpha \, \mathrm{d}s + \int_{\mathcal{A}} b_\alpha \bar{u}_\alpha \, \mathrm{d}A. \quad (3.129)$$

where the virtual pressure is taken as $\beta\bar{p}$ in order to preserve symmetry of the expression with respect to the pressure terms.

BIBLIOGRAPHY

Bathe, K. J., *Finite Element Procedures*, Prentice-Hall, Englewood Cliffs, NJ, 1996.

Brenner, S. C., and L. Ridgway Scott, *The Mathematical Theory of Finite Element Methods*, Springer, New York, 2008.

Brezzi, F., and M. Fortin, *Mixed and Hybrid Finite Element Methods*, Springer-Verlag, New York, 1991.

Hughes, T. J. R., *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*, Dover Publications, Mineola, NY, 2000.

Pian, T. H. H., and P. Tong, Basis of finite element methods for solid continua, *Int. J. Numer. Meth. Eng*., 1, 3–28, 1969.

Pian, T. H. H., and P. Tong, Mixed and hybrid finite-element methods, In *Finite Element Handbook*, Part 2, Chapter 5, H. Kardestuncer and D. H. Norrie, eds., McGraw-Hill, New York, 1987.

Wunderlich, W., Ein verallgemeinertes Variationsverfahren zur volen oder teilweisen Diskretisierung mehrdimensionaler Elastizitätsprobleme, *Ingenieur-Archiv*, 9, 230–247, 1970.

Zienkiewicz, O. C., R. L. Taylor, and J. Z. Zhu, *The Finite Element Method: Its Basis and Fundamentals*, 6th ed., Elsevier Butterworth-Heinemann, Burlington, MA, 2005.

Zienkiewicz, O. C., and R. L. Taylor, *The Finite Element Method for Solid and Structural Mechanics*, 6th ed., Elsevier Butterworth-Heinemann, Burlington, MA, 2005.

4 The Triangle and the Tetrahedron

4.1 LINEAR FUNCTIONS OVER A TRIANGULAR REGION

In order to complete the solution of the elasticity problem via the finite element method, we need to develop shape functions for some elements. For plane problems, it will be convenient to use triangular subregions of the x–y plane. Some tools are first collected here.

In calculations for triangular regions, it is often convenient to use special coordinates called triangular or area coordinates. For the triangular region ABC, shown in Figure 4.1, a point P is located by coordinate parameters ζ_1, ζ_2, and ζ_3. Coordinate ζ_1 is defined by

$$\zeta_1 = \frac{\text{length of } PQ}{\text{length of } AQ} = \frac{\text{area of } BPC}{\text{area of } BAC}. \tag{4.1}$$

Therefore, $\zeta_1 = 1$ at corner 1 and $\zeta_1 = 0$ at corner 2 and corner 3, and varies linearly with distance along each side. Similar definitions hold for ζ_2 and ζ_3. Note that the parameters are not independent since

$$\zeta_1 + \zeta_2 + \zeta_3 = 1 \tag{4.2}$$

for all points.

Denote the rectangular Cartesian coordinates of point A by (X_1, Y_1), B by (X_2, Y_2), and C by (X_3, Y_3). The triangular coordinates of A are (1,0,0). The equation of side BC is $\zeta_1 = 0$, and so forth. In general, for the x–y coordinates of any point, one finds

$$\begin{bmatrix} 1 \\ x \\ y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ X_1 & X_2 & X_3 \\ Y_1 & Y_2 & Y_3 \end{bmatrix} \begin{bmatrix} \zeta_1 \\ \zeta_2 \\ \zeta_3 \end{bmatrix} = \mathbf{\Phi}^{\mathrm{T}} \begin{bmatrix} \zeta_1 \\ \zeta_2 \\ \zeta_3 \end{bmatrix}, \tag{4.3}$$

where

$$\mathbf{\Phi} = \begin{bmatrix} 1 & X_1 & Y_1 \\ 1 & X_2 & Y_2 \\ 1 & X_3 & Y_3 \end{bmatrix}. \tag{4.4}$$

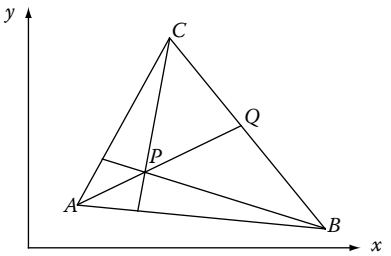


FIGURE 4.1 Triangular coordinates.

The inverse transformation is

$$\begin{bmatrix} \zeta_1 \\ \zeta_2 \\ \zeta_3 \end{bmatrix} = (\mathbf{\Phi}^{-1})^{\mathrm{T}} \begin{bmatrix} 1 \\ x \\ y \end{bmatrix}. \tag{4.5}$$

The inverse of the matrix $\mathbf{\Phi}$ is

$$\mathbf{\Phi}^{-1} = \frac{1}{2a} \begin{bmatrix} 2A_1 & 2A_2 & 2A_3 \\ a_1 & a_2 & a_3 \\ b_1 & b_2 & b_3 \end{bmatrix}, \tag{4.6}$$

where

$$\begin{array}{lll} 2A_1 = X_2Y_3 - X_3Y_2, & 2A_2 = X_3Y_1 - X_1Y_3, & 2A_3 = X_1Y_2 - X_2Y_1, \\ a_1 = Y_2 - Y_3, & a_2 = Y_3 - Y_1, & a_3 = Y_1 - Y_2, \\ b_1 = X_3 - X_2, & b_2 = X_1 - X_3, & b_3 = X_2 - X_1, \end{array} \tag{4.7}$$

and

$$a = \frac{1}{2} \begin{vmatrix} 1 & X_1 & Y_1 \\ 1 & X_2 & Y_2 \\ 1 & X_3 & Y_3 \end{vmatrix}. \tag{4.8}$$

The determinant is positive if the nodes are numbered counterclockwise, and a is the area of the triangle. The matrix Φ^{-1} depends only on the geometry of the triangle. Once the shape of the triangle is specified, Φ^{-1} can be calculated. It does not depend on the absolute position in space since only the differences in the nodal coordinates occur in 4.7.

One major importance of triangular coordinates is the identity

$$\iint \zeta_1^m \zeta_2^n \zeta_3^p \, dA = \frac{m!n!p!}{(m+n+p+2)!} 2a, \tag{4.9}$$

where m, n, and p are exponents, not indices. This makes integration of polynomials over the triangular region an easy task.

Now, consider the general linear function

$$f(x,y) = c_1 + c_2 x + c_3 y = \begin{bmatrix} 1 & x & y \end{bmatrix} \{c_i\}. \tag{4.10}$$

The values of f at points A, B, and C are

$$\begin{bmatrix} f_1 \\ f_2 \\ f_3 \end{bmatrix} = \begin{bmatrix} 1 & X_1 & Y_1 \\ 1 & X_2 & Y_2 \\ 1 & X_3 & Y_3 \end{bmatrix} \begin{bmatrix} c_1 \\ c_2 \\ c_3 \end{bmatrix} \text{ or } \{f_i\} = \mathbf{\Phi}\{c_i\}. \tag{4.11}$$

Therefore,

$$\{c_i\} = \mathbf{\Phi}^{-1}\{f_i\}. \tag{4.12}$$

Thus, function 4.10 can be expressed in terms of its values at the corners:

$$f(x,y) = [1 \; x \; y]\mathbf{\Phi}^{-1}\{f_i\}. \tag{4.13}$$

In the area coordinates, by 4.5,

$$f(x,y) = \zeta_1 f_1 + \zeta_2 f_2 + \zeta_3 f_3. \tag{4.14}$$

Thus, the general linear function $f(x,y)$ is expressed in a very simple way in terms of its values at the nodes by using the triangular coordinates.

We automatically achieve the task of constructing a continuous displacement field. To show this, let us consider a pair of triangles (Figure 4.2) and a general linear function over each one. Along side BC of triangle 1, we have $\zeta_1 = 0$ and

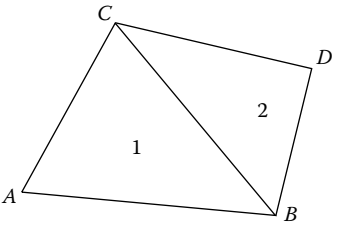


FIGURE 4.2 Two adjacent triangles.

$$f(0,\zeta_2,\zeta_3) = \zeta_2 f_2 + \zeta_3 f_3 = \zeta_2 f_2 + (1 - \zeta_2) f_3. \tag{4.15}$$

Thus, $f(x,y)$ varies linearly along the edge BC and values of f along the side depends only on values of f at the nodes lying on that side. If we consider another linear function defined in a similar manner over triangle 2 that shares the edge BC with triangle 1, we again find that f varies linearly from B to C and depends only on values at B and C. The two functions must be equal along line BC because both are linear and have the same values at B and C. Function f is therefore continuous across the element boundary, that is, ζ_i are shape functions for the three-node triangle.

By dividing any region into a number of such triangles, we can construct a continuous piecewise linear function over the region. Graphically, this function is an assembly of inclined planes joined above the edge of the triangles. In the limit, as the size of each triangle tends to zero, we can approximate any continuous function over the region as closely as we like by such piecewise linear functions. We can therefore use this representation to approximate each component of displacement in the plane problem. The fundamental unknowns then become the nodal values rather than the entire function. Instead of solving for a function, we have only to find a finite number of real numbers.

4.2 TRIANGULAR ELEMENT FOR PLANE STRESS AND PLANE STRAIN

The triangle has three corner nodes (Figure 4.3) with displacement components (U_i,V_i). The components u_1 and u_2 of displacement can be represented within a typical element by linear function 4.13:

$$\begin{aligned} u_1 &= [1\ x\ y]\,\mathbf{\Phi}^{-1}\mathbf{U} = \zeta_1 U_1 + \zeta_2 U_2 + \zeta_3 U_3, \\ u_2 &= [1\ x\ y]\,\mathbf{\Phi}^{-1}\mathbf{V} = \zeta_1 V_1 + \zeta_2 V_2 + \zeta_3 V_3, \end{aligned} \tag{4.16}$$

where

$$\begin{aligned} \mathbf{U} &= \begin{bmatrix} U_1 & U_2 & U_3 \end{bmatrix}^{\mathrm{T}}, \\ \mathbf{V} &= \begin{bmatrix} V_1 & V_2 & V_3 \end{bmatrix}^{\mathrm{T}}. \end{aligned} \tag{4.17}$$

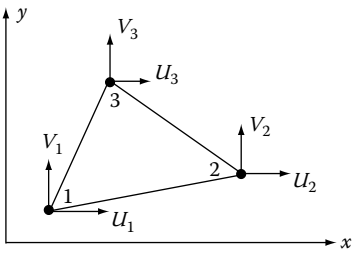


FIGURE 4.3 Triangular element with three nodes.

Differentiating with respect to x and y gives

$$u_{1,1} = [0\ 1\ 0]\,\boldsymbol{\Phi}^{-1}\mathbf{U} = \frac{1}{2a}\begin{bmatrix} a_1 & a_2 & a_3 \end{bmatrix}\mathbf{U}, \tag{4.18}$$

$$u_{1,2} = [0\ 0\ 1]\,\boldsymbol{\Phi}^{-1}\mathbf{U} = \frac{1}{2a}\begin{bmatrix} b_1 & b_2 & b_3 \end{bmatrix}\mathbf{U}, \tag{4.19}$$

$$u_{2,1} = [0\ 1\ 0]\,\boldsymbol{\Phi}^{-1}\mathbf{V} = \frac{1}{2a}\begin{bmatrix} a_1 & a_2 & a_3 \end{bmatrix}\mathbf{V}, \tag{4.20}$$

$$u_{2,2} = [0\ 0\ 1]\,\boldsymbol{\Phi}^{-1}\mathbf{V} = \frac{1}{2a}\begin{bmatrix} b_1 & b_2 & b_3 \end{bmatrix}\mathbf{V}. \tag{4.21}$$

U_k and V_k are the x and y components of displacement of the kth node. From 2.27 using 4.18 to 4.21 and 4.4, with $\gamma_{12} = 2\varepsilon_{12}$,

$$\boldsymbol{\varepsilon} = \mathbf{A}\mathbf{D} \tag{4.22}$$

where

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{bmatrix}, \quad \mathbf{D} = \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}, \tag{4.23}$$

and

$$\mathbf{A} = \frac{1}{2a}\begin{bmatrix} a_1 & a_2 & a_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & b_1 & b_2 & b_3 \\ b_1 & b_2 & b_3 & a_1 & a_2 & a_3 \end{bmatrix} = \frac{1}{2a}\begin{bmatrix} \lfloor a_k \rfloor & \mathbf{0} \\ \mathbf{0} & \lfloor b_k \rfloor \\ \lfloor b_k \rfloor & \lfloor a_k \rfloor \end{bmatrix}, \tag{4.24}$$

where a_i and b_i are given by 4.7.

For an isotropic material with no temperature change, from 2.81,

$$\boldsymbol{\tau} = \mathbf{C}\boldsymbol{\varepsilon}, \tag{4.25}$$

where

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{12} \end{bmatrix}. \tag{4.26}$$

For plane strain,

$$\mathbf{C} = \begin{bmatrix} 2\mu+\lambda & \lambda & 0 \\ \lambda & 2\mu+\lambda & 0 \\ 0 & 0 & \mu \end{bmatrix}. \tag{4.27}$$

For plane stress,

$$\mathbf{C} = \frac{2\mu}{1-\nu}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix}. \tag{4.28}$$

For the isotropic material in plane stress or plane strain, **C** has the form

$$\mathbf{C} = \begin{bmatrix} C_{11} & C_{12} & 0 \\ C_{12} & C_{11} & 0 \\ 0 & 0 & C_{33} \end{bmatrix}. \tag{4.29}$$

From 4.22 and 4.25,

$$\boldsymbol{\tau} = \mathbf{BD}, \tag{4.30}$$

where

$$\mathbf{B} = \mathbf{CA}. \tag{4.31}$$

From 3.24, applied to two dimensions, the element stiffness matrix is

$$\mathbf{k} = \int_{\mathcal{R}} \mathbf{B}^{\mathrm{T}}\mathbf{A}\,\mathrm{d}a. \tag{4.32}$$

Since **A** and **B** are constant, one finds

$$\mathbf{k} = a\mathbf{A}^{\mathrm{T}}\mathbf{CA} \tag{4.33}$$

where a is the area of the element. In this case, **k** is a 6×6 matrix:

$$\mathbf{k} = \begin{array}{c} \begin{matrix} \mathbf{U} & \mathbf{V} \end{matrix} \\ \begin{bmatrix} \mathbf{k}^{uu} & \mathbf{k}^{uv} \\ \mathbf{k}^{vu} & \mathbf{k}^{vv} \end{bmatrix} \end{array} \tag{4.34}$$

where

$$\begin{aligned}
\mathbf{k}^{uu} &= \frac{1}{4a}\Big[C_{11}a_k a_m + C_{33}b_k b_m\Big],\\
\mathbf{k}^{uv} &= \frac{1}{4a}\Big[C_{12}a_k b_m + C_{33}b_k a_m\Big],\\
\mathbf{k}^{vu} &= \frac{1}{4a}\Big[C_{12}b_k a_m + C_{33}a_k b_m\Big],\\
\mathbf{k}^{vv} &= \frac{1}{4a}\Big[C_{11}b_k b_m + C_{33}a_k a_m\Big].
\end{aligned} \tag{4.35}$$

The element nodal loads also have to be determined for given tractions. For example, consider a uniform load along the side 2–3, T_1^0 is constant and $T_2^0 = 0$. We need to evaluate the surface integral in 3.16, which becomes a line integral in two dimensions. In matrix notation,

$$\mathbf{f}^{\mathrm{T}}\mathbf{D} \equiv \int_c \mathbf{T}_0^{\mathrm{T}}\mathbf{u}\,\mathrm{d}s = \int_c T_i^0 u_i\,\mathrm{d}s = \int_c T_1^0 u_1\,\mathrm{d}s. \tag{4.36}$$

This is an identity in **D**. On side 2–3, $\zeta_1 = 0$, $\zeta_2 + \zeta_3 = 1$, and $\zeta_2 = 1 - s/L$, where s is the distance along side 2–3 from corner 2. Thus, on side 2, $\mathrm{d}s = -\mathrm{d}\zeta_2$, where L is the length of side 2–3. Therefore,

$$\begin{aligned}
\int T_1^0 u_1 \mathrm{d}s &= T_1^0 \int_{\zeta_2=1}^{0} [\zeta_1\ \ \zeta_2\ \ \zeta_3]\begin{bmatrix} U_1\\ U_2\\ U_3\end{bmatrix}\mathrm{d}s\\
&= LT_1^0 \int_{\zeta_2=0}^{1} [0\ \ \zeta_2\ \ (1-\zeta_2)]\mathbf{U}\mathrm{d}\zeta_2\\
&= LT_1^0\begin{bmatrix}0 & \frac{1}{2} & \frac{1}{2}\end{bmatrix}\mathbf{U} = LT_1^0\begin{bmatrix}\frac{1}{2} & \frac{1}{2}\end{bmatrix}\begin{bmatrix}U_2\\ U_3\end{bmatrix}.
\end{aligned} \tag{4.37}$$

Thus, the distributed load is equivalent to forces acting at adjacent nodes 2 and 3, each equal to one-half the total load. The other nodal forces are zero.

As we have seen in this section, it is often more convenient to use matrix notation in particular cases, rather than the multiple index notation. We will switch back and forth according to whichever notation is most convenient at the moment. Some care must be taken to distinguish between the index related to a component with respect to the basis of the coordinate system from an index that refers to a node number, element number, or merely to the position in a matrix array.

In the notation of 3.19, the shape functions corresponding to 4.16 and 4.17 are

$$
\begin{aligned}
N_{11} &= \frac{1}{2a}(2A_1 + a_1x + b_1y), & N_{21} &= 0\\
N_{12} &= 0, & N_{22} &= \frac{1}{2a}(2A_1 + a_1x + b_1y),\\
N_{13} &= \frac{1}{2a}(2A_2 + a_2x + b_2y), & N_{23} &= 0,\\
N_{14} &= 0, & N_{24} &= \frac{1}{2a}(2A_2 + a_2x + b_2y),\\
N_{15} &= \frac{1}{2a}(2A_3 + a_3x + b_3y), & N_{25} &= 0,\\
N_{16} &= 0, & N_{26} &= \frac{1}{2a}(2A_3 + a_3x + b_3y),
\end{aligned}
\tag{4.38}
$$

The geometric factors $A_{\alpha\beta k}$ defined by 3.4 can be read off from 4.38 as elements from matrix **A**. The coefficients $c_{\alpha\beta\gamma\delta}$ are elements of matrix **C**. The coefficients $B_{\alpha\beta k}$ are elements of matrix **B**. Elements of matrix **k** are exactly the k_{IJ}^m of 3.11, except that the explicit indication of element number by the index m has been omitted in this section.

4.3 PLANE QUADRILATERAL FROM FOUR TRIANGLES

A quadrilateral element is readily formed from four triangles with a common node at the centroid of the quadrilateral. The common node can then be eliminated if there is no body force.

Consider a typical quadrilateral region subdivided into four triangles as shown in Figure 4.4. Node number 5 can be placed at any interior point, but it is usually placed at the centroid of the quadrilateral and the coordinates (X_5,Y_5) are calculated from the coordinates of nodes 1–4.

The stiffness matrix for each triangle is given by 4.33. The 10 × 10 stiffness matrix of the complete quadrilateral is then formed by merging the element matrices as described in Section 4.4. Let us suppose that there are no forces associated with node 5. Then, Equation 3.18 for the quadrilateral becomes

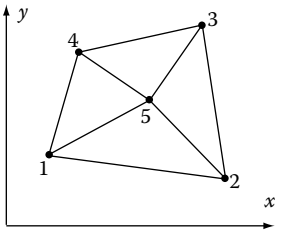


FIGURE 4.4 Four-triangle assembly.

$$\begin{bmatrix} \mathbf{k}_{11} & \mathbf{k}_{12} \\ \mathbf{k}_{21} & \mathbf{k}_{22} \end{bmatrix} \begin{bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \\ U_4 \\ V_4 \\ U_5 \\ V_5 \end{bmatrix} = \begin{bmatrix} FX_1 \\ FY_1 \\ FX_2 \\ FY_2 \\ FX_3 \\ FY_3 \\ FX_4 \\ FY_4 \\ 0 \\ 0 \end{bmatrix}, \tag{4.39}$$

where FX_k and FY_k are the node forces, and the stiffness matrix for the quadrilateral has been partitioned into four submatrices. The submatrix $\mathbf{k}_{22}$ is 2×2 and so forth. The last two equations can be used to solve for the displacements (U_5, V_5) of node 5:

$$\begin{bmatrix} U_5 \\ V_5 \end{bmatrix} = -\mathbf{k}_{22}^{-1}\mathbf{k}_{21}\mathbf{D}, \tag{4.40}$$

where the fundamental nodal displacements and nodal forces are

$$\begin{aligned} \mathbf{D} &= \begin{bmatrix} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 \end{bmatrix}^{\mathrm{T}}, \\ \mathbf{F} &= \begin{bmatrix} FX_1 & FY_1 & FX_2 & FY_2 & FX_3 & FY_3 & FX_4 & FY_4 \end{bmatrix}^{\mathrm{T}}. \end{aligned} \tag{4.41}$$

Next, use 4.40 to eliminate (U_5, V_5) from the first eight equations of 4.39 to obtain

$$\mathbf{KD} = \mathbf{F}, \tag{4.42}$$

where

$$\mathbf{K} = \mathbf{k}_{11} - \mathbf{k}_{12}\mathbf{k}_{22}^{-1}\mathbf{k}_{21}. \tag{4.43}$$

and $\mathbf{F}$ is the column matrix of forces acting at nodes 1 to 4. $\mathbf{K}$, an 8×8 matrix, is the stiffness matrix of the quadrilateral shown in Figure 4.5.

One may now use the quadrilateral as a basic finite element with a stiffness matrix given by 4.43. The region is divided into quadrilaterals (and possibly triangles also) and the global stiffness matrix is formed by merging the element

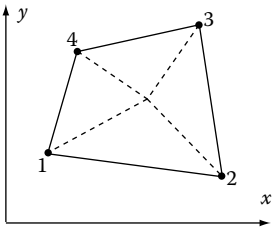


FIGURE 4.5 Quadrilateral formed from four triangles.

matrices. Note that the stress is still approximated as constant within each triangular subregion.

This process of elimination of unloaded internal nodes to obtain a stiffness matrix for a patch of elements is called *static condensation*. The patch of elements can then be used as a building block for a finite element model, just as one uses a single element.

Static condensation may be used in other ways. For example, shape functions are sometimes used in conjunction with parameters that are not interpreted as nodal values of the displacement. Such nodeless parameters may then be eliminated by static condensation to obtain the stiffness matrix for the element.

4.3.1 Square Element Formed from Four Triangles

The nodes and elements are numbered as shown in Figure 4.6. The nodal coordinates for each element to be used in Equation 4.7 are shown in Table 4.1. The area of each element is $h^2/4$. The geometric parameters in 4.24 are shown in Table 4.2.

For $\nu = 1/3$, $\mu = 3E/8$, from 4.28 the matrix $\mathbf{C}$ for plane stress is

$$\mathbf{C} = \mu \begin{bmatrix} 3 & 1 & 0 \\ 1 & 3 & 0 \\ 0 & 0 & 1 \end{bmatrix}. \tag{4.44}$$

The stiffness matrix for each element is calculated from 4.33:

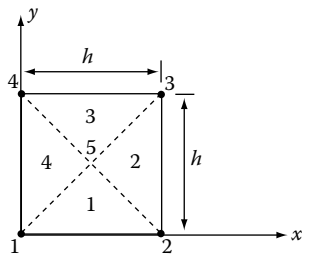


FIGURE 4.6 Square from four triangles.

TABLE 4.1
Element Geometry

Element	Nodes	X_1/h	X_2/h	X_3/h	Y_1/h	Y_2/h	Y_3/h
1	1–2–5	0	1	1/2	0	0	1/2
2	2–3–5	1	1	1/2	0	1	1/2
3	3–4–5	1	0	1/2	1	1	1/2
4	4–1–5	0	0	1/2	1	0	1/2

$$\mathbf{k}^1 = \frac{\mu}{4}\begin{array}{c} \begin{matrix} U_1 & V_1 & U_2 & V_2 & U_5 & V_5 \end{matrix} \\ \begin{bmatrix} 4 & 2 & -2 & 0 & -2 & -2 \\ 2 & 4 & 0 & 2 & -2 & -6 \\ -2 & 0 & 4 & -2 & -2 & 2 \\ 0 & 2 & -2 & 4 & 2 & -6 \\ -2 & -2 & -2 & 2 & 4 & 0 \\ -2 & -6 & 2 & -6 & 0 & 12 \end{bmatrix} \end{array}, \quad \mathbf{k}^3 = \frac{\mu}{4}\begin{array}{c} \begin{matrix} U_3 & V_3 & U_4 & V_4 & U_5 & V_5 \end{matrix} \\ \begin{bmatrix} 4 & 2 & -2 & 0 & -2 & -2 \\ 2 & 4 & 0 & 2 & -2 & -6 \\ -2 & 0 & 4 & -2 & -2 & 2 \\ 0 & 2 & -2 & 4 & 2 & -6 \\ -2 & -2 & -2 & 2 & 4 & 0 \\ -2 & -6 & 2 & -6 & 0 & 12 \end{bmatrix} \end{array}, \tag{4.45}$$

$$\mathbf{k}^2 = \frac{\mu}{4}\begin{array}{c} \begin{matrix} U_2 & V_2 & U_3 & V_3 & U_5 & V_5 \end{matrix} \\ \begin{bmatrix} 4 & -2 & 2 & 0 & -6 & 2 \\ -2 & 4 & 0 & -2 & 2 & -2 \\ 2 & 0 & 4 & 2 & -6 & -2 \\ 0 & -2 & 2 & 4 & -2 & -2 \\ -6 & 2 & -6 & -2 & 12 & 0 \\ 2 & -2 & -2 & -2 & 0 & 4 \end{bmatrix} \end{array}, \quad \mathbf{k}^4 = \frac{\mu}{4}\begin{array}{c} \begin{matrix} U_4 & V_4 & U_1 & V_1 & U_5 & V_5 \end{matrix} \\ \begin{bmatrix} 4 & -2 & 2 & 0 & -6 & 2 \\ -2 & 4 & 0 & -2 & 2 & -2 \\ 2 & 0 & 4 & 2 & -6 & -2 \\ 0 & -2 & 2 & 4 & -2 & -2 \\ -6 & 2 & -6 & -2 & 12 & 0 \\ 2 & -2 & -2 & -2 & 0 & 4 \end{bmatrix} \end{array}. \tag{4.46}$$

TABLE 4.2
Element Parameters

Element	Nodes	a_1/h	a_2/h	a_3/h	b_1/h	b_2/h	b_3/h
1	1–2–5	−1/2	+1/2	0	−1/2	−1/2	+1
2	2–3–5	+1/2	+1/2	−1	−1/2	+1/2	0
3	3–4–5	+1/2	−1/2	0	+1/2	+1/2	−1
4	4–1–5	−1/2	−1/2	+1	+1/2	−1/2	0

The element matrices are merged to form the global matrix:

$$\mathbf{k} = \frac{\mu}{4}\begin{array}{c} \begin{matrix} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 & U_5 & V_5 \end{matrix} \\ \begin{bmatrix} 8 & 4 & -2 & 0 & 0 & 0 & 2 & 0 & -8 & -4 \\ 4 & 8 & 0 & 2 & 0 & 0 & 0 & -2 & -4 & -8 \\ -2 & 0 & 8 & -4 & 2 & 0 & 0 & 0 & -8 & -4 \\ 0 & 2 & -4 & 8 & 0 & -2 & 0 & 0 & 4 & -8 \\ 0 & 0 & 2 & 0 & 8 & 4 & -2 & 0 & -8 & -4 \\ 0 & 0 & 0 & -2 & 4 & 8 & 0 & 2 & -4 & -8 \\ 2 & 0 & 0 & 0 & -2 & 0 & 8 & -4 & -8 & 4 \\ 0 & -2 & 0 & 0 & 0 & 2 & -4 & 8 & 4 & -8 \\ -8 & -4 & -8 & 4 & -8 & -4 & -8 & 4 & 32 & 0 \\ -4 & -8 & 4 & -8 & -4 & -8 & 4 & -8 & 0 & 32 \end{bmatrix} \end{array}. \quad (4.47)$$

We can eliminate node 5 by static condensation:

$$\mathbf{k}_{22} = 8\mu\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \quad \mathbf{k}_{22}^{-1} = \frac{1}{8\mu}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}, \quad (4.48)$$

$$\mathbf{k}_{12}^{\mathrm{T}} = \mathbf{k}_{21} = \mu\begin{bmatrix} -2 & -1 & -2 & 1 & -2 & -1 & -2 & 1 \\ -1 & -2 & 1 & -2 & -1 & -2 & 1 & -2 \end{bmatrix}. \quad (4.49)$$

From (4.43), the stiffness matrix for the square is

$$\mathbf{K} = \frac{\mu}{8}\begin{array}{c} \begin{matrix} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 \end{matrix} \\ \begin{bmatrix} 11 & 4 & -7 & 0 & -5 & -4 & 1 & 0 \\ 4 & 11 & 0 & 1 & -4 & -5 & 0 & -7 \\ -7 & 0 & 11 & -4 & 1 & 0 & -5 & 4 \\ 0 & 1 & -4 & 11 & 0 & -7 & 4 & -5 \\ -5 & -4 & 1 & 0 & 11 & 4 & -7 & 0 \\ -4 & -5 & 0 & -7 & 4 & 11 & 0 & 1 \\ 1 & 0 & -5 & 4 & -7 & 0 & 11 & -4 \\ 0 & -7 & 4 & -5 & 0 & 1 & -4 & 11 \end{bmatrix} \end{array}. \quad (4.50)$$

This will be compared with the direct generation of the stiffness matrix for the quadrilateral in the next chapter.

4.4 PLANE STRESS EXAMPLE: SHORT BEAM

The plane stress problem described in Section 2.1.14, illustrated in Figure 4.7, will now be solved by using finite elements that are triangles with constant strain.

The solution of this problem is of the form

$$\begin{aligned} \tau_{\alpha\beta} &= pS_{\alpha\beta}, \\ u_\alpha &= \frac{pa}{E}\bar{u}_\alpha. \end{aligned} \tag{4.51}$$

where $S_{\alpha\beta}$, $\bar{u}_\alpha$ are the stress tensor and displacement vector for unit side length, unit modulus, and unit load p. The following solution will be for these nondimensional variables. However, the true value of ν must be used.

Use can be made of the antisymmetrical pattern of deformation. The displacements are such that

$$\begin{aligned} u(x, y) &= u(-x, y), \\ v(x, y) &= -v(-x, y). \end{aligned} \tag{4.52}$$

Therefore,

$$\begin{aligned} \varepsilon_{11}(x, y) &= -\varepsilon_{11}(-x, y), \\ \varepsilon_{22}(x, y) &= -\varepsilon_{22}(-x, y), \\ \varepsilon_{12}(x, y) &= +\varepsilon_{12}(-x, y). \end{aligned} \tag{4.53}$$

And

$$\begin{aligned} \tau_{11}(x, y) &= -\tau_{11}(-x, y), \\ \tau_{22}(x, y) &= -\tau_{22}(-x, y), \\ \tau_{12}(x, y) &= +\tau_{12}(-x, y). \end{aligned} \tag{4.54}$$

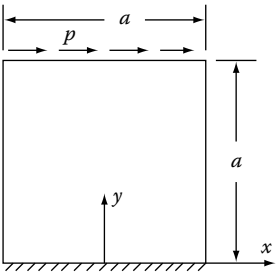


FIGURE 4.7 Short beam.

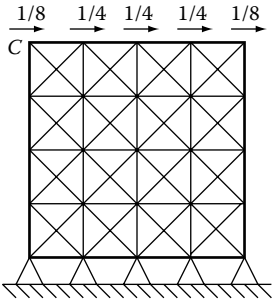


FIGURE 4.8 Element layout.

Along the line $x = 0$, we have the antisymmetry conditions: $u_2 = 0$, $\tau_{11} = \tau_{22} = 0$. Therefore, we only need to consider one-half of the region and use the symmetry conditions as boundary conditions. However, in the following example analysis, the entire region is retained for illustration.

A typical element layout is shown in Figure 4.8. In general, there are N segments per side. The nodes on the bottom edge have zero displacement. The generalized forces for nodes on the top edge are $(0, 1/N)$, except for the corner nodes where they are 1/2 as much. Other generalized forces are zero. The calculated x displacement of corner C is shown in Table 4.2 for $\nu = 1/3$.

The calculated displacement is always too small but increases monotonically to the true solution as the number of elements increases. Figure 4.9 shows the calculated displacement as a function of the number of divisions in this layout.

The stress varies smoothly in the exact solution. However, in the finite element model, the strains and therefore the stresses are constant throughout each triangle. Some interpolation of the calculated values is therefore necessary. A common method of interpreting the finite element analysis is to take the calculated values to

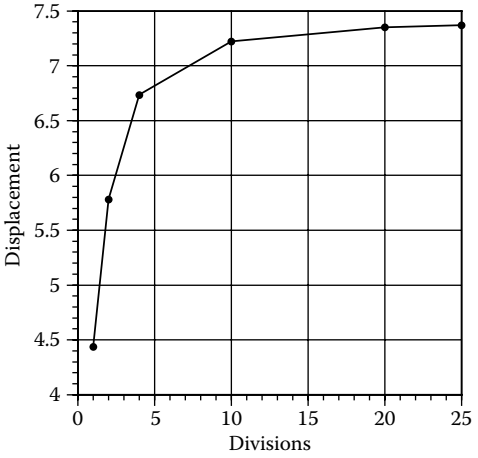


FIGURE 4.9 Convergence toward the exact solution.

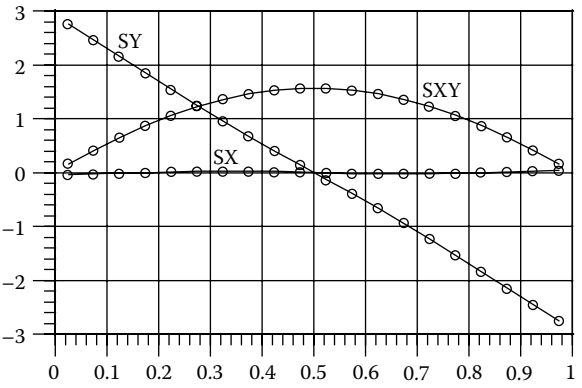


FIGURE 4.10 Stress at $y = 0.5083$.

be the stress state at the centroid of the triangle. Figures 4.10 through 4.13 show the results for 1600 triangles using that interpretation.

The exact solution to this simple-looking problem is unknown. The stress state distant from the top and bottom edges was estimated in Section 2.1.14, Equations 2.52 and 2.53. Looking at Figure 4.10, one sees very good agreement with this elementary theory on the interior of the region.

As expected, Figure 4.11 shows that the elementary theory is not satisfactory near the support where the stress distribution depends strongly on the details of the support condition. Figure 4.12 shows that the elementary theory is also not satisfactory near the loaded edge, where the stress distribution depends strongly on the details of the load distribution.

We have analyzed the situation in the corner $x = 0$, $y = 0$, as given by 2.58. We expect a singularity (infinite stress) at the corners where the free and fixed boundaries join. The effect of the singularity can be seen in Figures 4.11 and 4.13. The infinite stresses at those points are, of course, never found via a numerical analysis. For each finer element mesh, one simply finds higher and higher stresses near the corners.

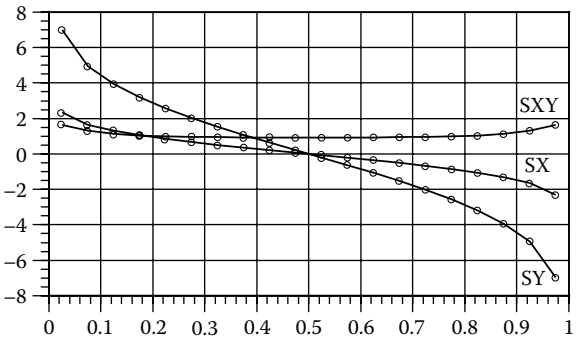


FIGURE 4.11 Stress at $y = 0.0083$.

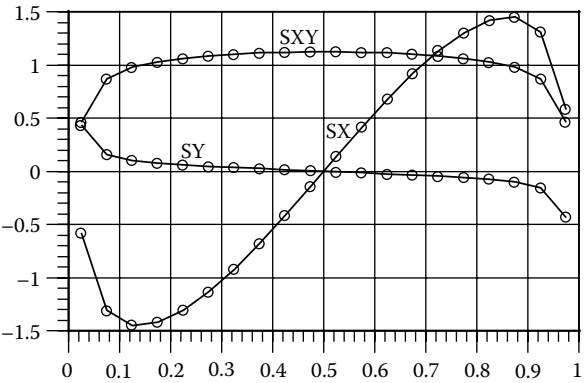


FIGURE 4.12 Stress at $y = 0.9917$.

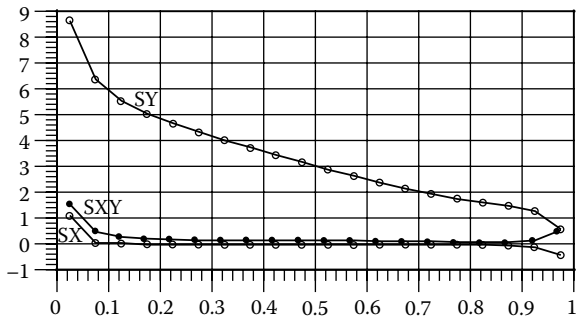


FIGURE 4.13 Stress at $x = 0.0083$.

4.4.1 Extrapolation of the Solution

Let us consider the solution at some fixed point (x_0,y_0) within an element. The exact solution has the power series expansion

$$u(x,y) = u(x_0,y_0) + u_x(x_0,y_0)(x - x_0) + u_y(x_0,y_0)(y - y_0) + \cdots, \tag{4.55}$$

where the subscripts denote the partial derivatives. Within the element, we have $x - x_0 \le h$ and $y - y_0 \le h$, where h is a characteristic dimension of the element. For the exact solution,

$$u \equiv u(h,h) = u^0 + u_x^0 h + u_y^0 h + kh^2 + O(h^3). \tag{4.56}$$

The finite element approximation is $\bar{u}$, which is obtained by retaining only the linear terms in the series representation:

$$\bar{u} = u^0 + u_x^0 h + u_y^0 h. \tag{4.57}$$

TABLE 4.3
Convergence

Segments per Side	No. of Nodes	No. of Triangles	No. of Rectangles	Corner Displacement
1	5	4	1	4.444
2	13	16	4	5.785
4	41	64	16	6.735
10	221	400	100	7.223
20	841	1600	400	7.350
25	1301	2500	625	7.372

The finite element approximation therefore differs from the exact solution by

$$u - \bar{u} = kh^2 + O(h^3). \tag{4.58}$$

Let h be a small number and denote the finite element solution for some value h_1 by $\bar{u}_1$, and by $\bar{u}_2$ for h_2. Neglecting the higher-order terms,

$$\frac{u - \bar{u}_2}{u - \bar{u}_1} \doteq \alpha^2, \quad \alpha = \frac{h_2}{h_1}. \tag{4.59}$$

Thus,

$$u \doteq \frac{\bar{u}_2 - \alpha^2 \bar{u}_1}{1 - \alpha^2}. \tag{4.60}$$

This formula can be used to estimate the exact solution from the finite element calculations. From the data in Table 4.3, we have the following estimate for $\bar{u}_2 = 7.372$, $\bar{u}_1 = 7.350$, and $\alpha = 20/25 = 0.8$ is

$$u \doteq \frac{7.372 - (0.8)^2 7.350}{1 - (0.8)^2} = 7.41. \tag{4.61}$$

We will see later from a more detailed finite element solution that the exact answer is still larger.

4.5 LINEAR STRAIN TRIANGLES

We have seen in Section 4.1 how a general linear expression for displacement led naturally to a triangular with three corner nodes. The strains in that triangle are constant. Let us now consider a general quadratic expression for displacement. We will see that the quadratic expression leads naturally to a six-node triangle.

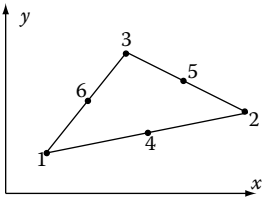


FIGURE 4.14 Linear strain triangle.

Suppose that each component of displacement is represented in the form

$$f(x,y) = c_1 + c_2x + c_3y + c_4x^2 + c_5xy + c_6y^2. \tag{4.62}$$

There are six constants that can be expressed in terms of the value of the function at six points. We will choose the three corners and the midpoints of the sides of the triangle (Figure 4.14). The constants could then be found by using these six conditions to solve for the six constants in terms of the nodal coordinates. However, all of that algebra can be avoided by proceeding directly to shape functions determined by trial and error. One can easily see that possible quadratic shape functions are as follows:

$$\begin{aligned} N_1 &= \zeta_1(2\zeta_1 - 1), & N_4 &= 4\zeta_1\zeta_2, \\ N_2 &= \zeta_2(2\zeta_2 - 1), & N_5 &= 4\zeta_2\zeta_3, \\ N_3 &= \zeta_3(2\zeta_3 - 1), & N_6 &= 4\zeta_3\zeta_1. \end{aligned} \tag{4.63}$$

These shape functions have the necessary property that $N_i = 1$ at the ith node and zero at all others. Along a side, say 1–4–2, where $\zeta_3 = 0$ and $\zeta_2 = 1 - \zeta_1$, we have

$$u = \zeta_1(2\zeta_1 - 1)U_1 + (1 - \zeta_1)(1 - 2\zeta_1)U_2 + 4\zeta_1(1 - \zeta_1)U_4. \tag{4.64}$$

That is, the displacement component is a quadratic function that is uniquely determined by the value of the displacement at the three nodes. An adjacent element that shares those three nodes will therefore have matching displacements along that edge. We conclude that the shape functions generate a continuous displacement field that is capable of exact representation of constant strain states. Calculation of the stiffness matrix is straightforward. Integration over the triangular region is easy because of the general formula for integration of polynomials 4.9.

4.6 FOUR-NODE TETRAHEDRON

The elements used for plane problems can easily be extended to three-dimensional regions. The three-node constant strain triangle for two dimensions can be directly extended to a four-node tetrahedron (Figure 4.15) for three dimensions by assuming a linear expression in (x, y, z) for the three components of displacement. There are now four natural coordinates ζ_i defined by the ratio of distance from the corner to the opposite side. The shape functions are $N_i = \zeta_i$.

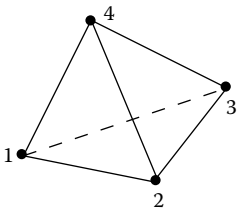


FIGURE 4.15 The four-node tetrahedron.

4.7 TEN-NODE TETRAHEDRON

The 2D triangle with mid-side nodes becomes the 10-node tetrahedron in three dimensions (Figure 4.16).

The shape functions are easily generated:

nodes 1, 2, 3, 4: $N_i = \zeta_i(2\zeta_i - 1)$
nodes 5, 6, 7: $N_5 = \zeta_1\zeta_2$, $N_6 = \zeta_2\zeta_3$, $N_7 = \zeta_3\zeta_1$
nodes 8.9.10: $N_8 = \zeta_1\zeta_4$, $N_9 = \zeta_2\zeta_4$, $N_{10} = \zeta_3\zeta_4$

See ANSYS elements Solid 92 and 187.

4.8 PROBLEMS

1. For the example problem shown in Figure 4.8, determine the force matrix **F** for a layout of triangles with $N = 3$ segments per side. You can label the elements and nodes in any convenient way.
2. Derive the nodal forces for a uniform body force $b_1 = 1$ for a three-node triangular element in plane stress. Show your answer on a diagram.
3. For the triangle element with linearly varying load T_1 as shown in Figure 4.17, determine the element force matrix.
4. Consider the layout of triangular elements for plane stress with eight elements and nine nodes as shown in Figure 4.18.

   Mark an X for the terms that may be nonzero and a 0 for the terms that must be zero in the first row of the global stiffness matrix.

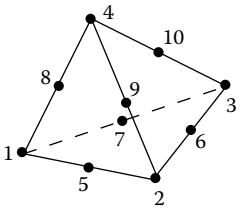


FIGURE 4.16 The 10-node tetrahedron.

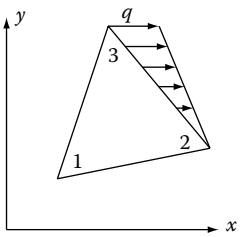


FIGURE 4.17 Linearly varying load.

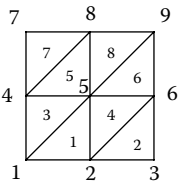


FIGURE 4.18 Eight-element model.

5. Consider the plane stress problem shown in Figure 4.19 with $\nu = 1/3$.
   (a) Determine the stiffness matrix for element 2.
   (b) Merge that stiffness matrix into the global stiffness matrix.
6. For the plane stress problem shown in Figure 4.19,
   (a) Determine the 10×10 global stiffness matrix.
   (b) Eliminate node 3 by static condensation to obtain an 8×8 stiffness matrix for the rectangle.
   (c) Solve for the displacements at nodes 4 and 5, when both components of displacement at nodes 1 and 2 are zero, and there is a uniformly distributed horizontal load p on side 4–5.

   Answers:

$$\overset{10\times10}{\mathbf{K}} = \frac{\mu}{4}\begin{array}{c} \begin{matrix} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 & U_5 & V_5 \end{matrix} \\ \begin{bmatrix} 8 & 4 & -2 & 0 & 0 & 0 & 2 & 0 & -8 & -4 \\ & 8 & 0 & 2 & 0 & 0 & 0 & -2 & -4 & -8 \\ & & 8 & -4 & 2 & 0 & 0 & 0 & -8 & 4 \\ & & & 8 & 0 & -2 & 0 & 0 & 4 & -8 \\ & & & & 8 & 4 & -2 & 0 & -8 & -4 \\ & & & & & 8 & 0 & 2 & -4 & -8 \\ & & & & & & 8 & -4 & -8 & 4 \\ & & & & & & & 8 & 4 & -8 \\ & & & & & & & & 32 & 0 \\ & & & & & & & & & 32 \end{bmatrix} \end{array}$$

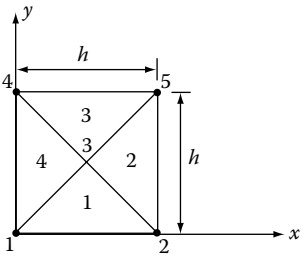


FIGURE 4.19 Four-element layout.

7. Verify that the ANSYS element 182 correctly models the constant strain triangle (Figure 4.20) by using ANSYS to determine the first column of the element stiffness matrix for the element shown. Use $E = 16$, $\nu = 1/3$. Compare the result with $\mathbf{k}^1$ from 4.45. (See Section 15.6.)
8. Solve problem 6 using the ANSYS program. Submit a list of nodal displacements. Note: To input nodes for a triangle, the third node is entered twice as the required fourth node for the Plane182 element. (See Section 15.7.)
9. Solve the short beam example by ANSYS using the element layout of Figure 4.8 consisting of all triangles. Determine the horizontal displacement of the upper right corner and compare with Table 4.3. (See Section 15.7.)

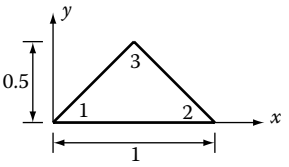


FIGURE 4.20 One triangle.

5 The Quadrilateral and the Hexahedron

5.1 FOUR-NODE PLANE RECTANGLE

For an arbitrary rectangle (Figure 5.1), choose the x–y axes as shown and number the corners as shown. We now wish to approximate a function $f(x,y)$ in this region. If the function varies linearly along an edge, it is uniquely determined along that edge by its values at the corners and continuity between elements that share those corners will be achieved. This will be the case if

$$f(x,y) = c_1 + c_2 x + c_3 y + c_4 xy = \begin{bmatrix} 1 & x & y & xy \end{bmatrix} \{c_k\}, \tag{5.1}$$

since the function is linear on an edge where either x or y is constant. The four coefficients can be expressed in terms of the four nodal values of the function, $f_k = f(X_k,Y_k)$, where $X_k = \pm a/2$ and $Y_k = \pm b/2$ are the coordinates of the kth node:

$$\{f_k\} = \mathbf{L}\{c_k\} \tag{5.2}$$

where

$$\mathbf{L} = \begin{bmatrix} 1 & -\frac{a}{2} & -\frac{b}{2} & +\frac{ab}{4} \\ 1 & +\frac{a}{2} & -\frac{b}{2} & -\frac{ab}{4} \\ 1 & +\frac{a}{2} & +\frac{b}{2} & +\frac{ab}{4} \\ 1 & -\frac{a}{2} & +\frac{b}{2} & -\frac{ab}{4} \end{bmatrix}. \tag{5.3}$$

The inverse of $\mathbf{L}$ is

$$\mathbf{L}^{-1} = \frac{1}{4}\begin{bmatrix} 1 & 1 & 1 & 1 \\ -\frac{2}{a} & +\frac{2}{a} & +\frac{2}{a} & -\frac{2}{a} \\ -\frac{2}{b} & -\frac{2}{b} & +\frac{2}{b} & +\frac{2}{b} \\ +\frac{4}{ab} & -\frac{4}{ab} & +\frac{4}{ab} & -\frac{4}{ab} \end{bmatrix}. \tag{5.4}$$

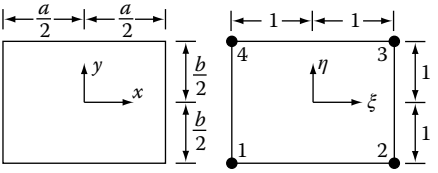


FIGURE 5.1 Four-node element.

Therefore,

$$f(x,y)=\begin{bmatrix}1 & x & y & xy\end{bmatrix}\mathbf{L}^{-1}\{f_k\}=\mathbf{N}\{f_k\} \tag{5.5}$$

where

$$\mathbf{N}=\frac{1}{4}\begin{bmatrix}1-\dfrac{2x}{a}-\dfrac{2y}{b}+\dfrac{4xy}{ab}\\ 1+\dfrac{2x}{a}-\dfrac{2y}{b}-\dfrac{4xy}{ab}\\ 1+\dfrac{2x}{a}+\dfrac{2y}{b}+\dfrac{4xy}{ab}\\ 1-\dfrac{2x}{a}+\dfrac{2y}{b}-\dfrac{4xy}{ab}\end{bmatrix}. \tag{5.6}$$

It is convenient to use the nondimensional coordinates

$$\xi=\frac{2x}{a},\quad \eta=\frac{2y}{b},\ -1\le\xi\le 1,\quad -1\le\eta\le 1. \tag{5.7}$$

Then,

$$\begin{aligned}N_1&\equiv\frac{1}{4}(1-\xi)(1-\eta),\ \ N_2\equiv\frac{1}{4}(1+\xi)(1-\eta),\\ N_3&\equiv\frac{1}{4}(1+\xi)(1+\eta),\ \ N_4\equiv\frac{1}{4}(1-\xi)(1+\eta),\end{aligned} \tag{5.8}$$

and

$$f(x,y)=N_1(\xi,\eta)f_1+N_2(\xi,\eta)f_2+N_3(\xi,\eta)f_3+N_4(\xi,\eta)f_4=\sum_{k=1}^{4}N_k f_k. \tag{5.9}$$

The formulas for N_k can be written concisely as

$$N_k = \frac{1}{4}(1+\xi_k\xi)(1+\eta_k\eta) \tag{5.10}$$

where ξ_k and η_k are the coordinates of the nodes in the $\xi-\eta$ system. The functions N_k are called *shape functions* or interpolation functions. They have the property

$$N_k(\xi_m,\eta_m) = \delta_{km}. \tag{5.11}$$

The functions in 5.10 are known as the *serendipity* functions because they could have been easily found by trial and error. Recall that any functions that satisfy the following three requirements may be used as shape functions:

(1) The shape function for a degree of freedom (DOF) must have the value of 1 for that DOF and 0 for all others: $N_k(\xi_m,\eta_m) = \delta_{km}$.
(2) The displacement field generated by the shape functions must be continuous between elements.
(3) The representation of displacement within an element must contain the lower-order terms of the power series representation for the function so that arbitrary constant strain states can be represented.

It does not matter how we discover the functions. We will subsequently generate other elements by a judicious choice of the shape functions.

We will need the derivatives of the shape functions:

$$\frac{\partial N_k}{\partial \xi} = \frac{1}{4}\xi_k(1+\eta_k\eta), \quad \frac{\partial N_k}{\partial \eta} = \frac{1}{4}\eta_k(1+\xi_k\xi). \tag{5.12}$$

Hence,

$$\begin{aligned} \frac{\partial N_k}{\partial x} &= \frac{\partial N_k}{\partial \xi}\frac{\mathrm{d}\xi}{\mathrm{d}x} = \frac{1}{2a}\xi_k(1+\eta_k\eta) \equiv A_k^1, \\ \frac{\partial N_k}{\partial y} &= \frac{\partial N_k}{\partial \eta}\frac{\mathrm{d}\eta}{\mathrm{d}y} = \frac{1}{2b}\eta_k(1+\xi_k\xi) \equiv A_k^2. \end{aligned} \tag{5.13}$$

Therefore,

$$\begin{aligned} \frac{\partial f}{\partial x} &= \sum_{k=1}^{4}\frac{\partial N_k}{\partial x}f_k = \sum_{k=1}^{4}\frac{1}{2a}\xi_k(1+\eta_k\eta)f_k = \mathbf{A}_1\left\{f_k\right\}, \\ \frac{\partial f}{\partial y} &= \sum_{k=1}^{4}\frac{\partial N_k}{\partial y}f_k = \sum_{k=1}^{4}\frac{1}{2b}\eta_k(1+\xi_k\xi)f_k = \mathbf{A}_2\left\{f_k\right\}, \end{aligned} \tag{5.14}$$

where

$$\mathbf{A}_1 = \left\lfloor A_k^1 \right\rfloor = \left\lfloor \frac{1}{2a}\xi_k(1+\eta_k\eta) \right\rfloor,$$
$$\mathbf{A}_2 = \left\lfloor A_k^2 \right\rfloor = \left\lfloor \frac{1}{2b}\eta_k(1+\xi_k\xi) \right\rfloor. \tag{5.15}$$

For a plane region, we approximate $u(x,y)$ and $v(x,y)$ by 5.9:

$$u = \sum_{k=1}^{4} N_k U_k, \quad v = \sum_{k=1}^{4} N_k V_k, \tag{5.16}$$

where U_k denotes the value of $u(x,y)$ at corner k, whereas V_k denotes the value of $v(x,y)$ at corner k. That is,

$$\mathbf{u} = \mathbf{ND} \tag{5.17}$$

where

$$\mathbf{u} = \begin{bmatrix} u \\ v \end{bmatrix}, \quad \mathbf{N} = \begin{bmatrix} \lfloor N_k \rfloor & \mathbf{0} \\ \mathbf{0} & \lfloor N_k \rfloor \end{bmatrix}, \quad \mathbf{D} = \begin{bmatrix} U_k \\ V_k \end{bmatrix}. \tag{5.18}$$

From 2.1, 5.14, and 5.16,

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{bmatrix} = \mathbf{AD} \tag{5.19}$$

where

$$\mathbf{A} = \begin{bmatrix} \mathbf{A}_1 & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_2 \\ \mathbf{A}_2 & \mathbf{A}_1 \end{bmatrix}, \quad \mathbf{A}_1 = \left\lfloor A_k^1 \right\rfloor, \quad \mathbf{A}_2 = \left\lfloor A_k^2 \right\rfloor. \tag{5.20}$$

Equation 5.19 is of the same form as 4.22, but here $\mathbf{A}$ is a 3×8 matrix since there are eight displacement parameters, and the components of $\mathbf{A}$ are not constant. From 3.24, the stiffness matrix for the plane rectangle is

$$\mathbf{k} = \iint \mathbf{A}^{\mathrm{T}}\mathbf{C}\mathbf{A}\,\mathrm{d}x\,\mathrm{d}y = \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1} \mathbf{A}^{\mathrm{T}}\mathbf{C}\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta\,. \tag{5.21}$$

For an isotropic elastic material,

$$\mathbf{C} = \begin{bmatrix} C_{11} & C_{12} & 0 \\ C_{12} & C_{11} & 0 \\ 0 & 0 & C_{33} \end{bmatrix}. \tag{5.22}$$

For plane stress, $C_{11} = \dfrac{E}{1-\nu^2}$, $C_{12} = \dfrac{\nu E}{1-\nu^2}$, $C_{33} = \dfrac{E}{1-\nu^2}\dfrac{1-\nu}{2}$. For plane strain, $C_{11} = \lambda + 2\mu$, $C_{12} = \lambda$, $C_{33} = \mu$. Substituting 5.20 and 5.22 into 5.21, we find

$$\begin{aligned} \mathbf{k} &= \frac{ab}{4}\iint \begin{bmatrix} \mathbf{A}_1^{\mathrm{T}} & \mathbf{0} & \mathbf{A}_2^{\mathrm{T}} \\ \mathbf{0} & \mathbf{A}_2^{\mathrm{T}} & \mathbf{A}_1^{\mathrm{T}} \end{bmatrix} \begin{bmatrix} C_{11} & C_{12} & 0 \\ C_{21} & C_{11} & 0 \\ 0 & 0 & C_{33} \end{bmatrix} \begin{bmatrix} \mathbf{A}_1 & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_2 \\ \mathbf{A}_2 & \mathbf{A}_1 \end{bmatrix} \mathrm{d}\xi\mathrm{d}\eta \\ &= \frac{ab}{4}\iint \begin{bmatrix} C_{11}\mathbf{A}_1^{\mathrm{T}}\mathbf{A}_1 + C_{33}\mathbf{A}_2^{\mathrm{T}}\mathbf{A}_2 & C_{12}\mathbf{A}_1^{\mathrm{T}}\mathbf{A}_2 + C_{33}\mathbf{A}_2^{\mathrm{T}}\mathbf{A}_1 \\ C_{21}\mathbf{A}_2^{\mathrm{T}}\mathbf{A}_1 + C_{33}\mathbf{A}_1^{\mathrm{T}}\mathbf{A}_2 & C_{11}\mathbf{A}_2^{\mathrm{T}}\mathbf{A}_2 + C_{33}\mathbf{A}_1^{\mathrm{T}}\mathbf{A}_1 \end{bmatrix} \mathrm{d}\xi\mathrm{d}\eta. \end{aligned} \tag{5.23}$$

From 5.13

$$\begin{aligned} \mathbf{A}_1^{\mathrm{T}}\mathbf{A}_1 &= \left[A_k^1 A_m^1\right] = \frac{1}{4a^2}\left[\xi_k\xi_m\left(1+\eta_k\eta+\eta_m\eta+\eta_k\eta_m\eta^2\right)\right], \\ \mathbf{A}_1^{\mathrm{T}}\mathbf{A}_2 &= \left[A_k^1 A_m^2\right] = \frac{1}{4ab}\left[\xi_k\eta_m\left(1+\xi_m\xi+\eta_k\eta+\xi_m\eta_k\xi\eta\right)\right], \\ \int_{-1}^{1}\int_{-1}^{1} A_k^1 A_m^1\,\mathrm{d}\xi\,\mathrm{d}\eta &= \frac{1}{4a^2}\left(\xi_k\xi_m\left(4+\frac{4}{3}\eta_k\eta_m\right)\right), \\ \int_{-1}^{1}\int_{-1}^{1} A_k^1 A_m^2\,\mathrm{d}\xi\,\mathrm{d}\eta &= \frac{1}{ab}\xi_k\eta_m, \text{ etc.} \end{aligned} \tag{5.24}$$

Integrating 5.23 term by term, we obtain

$$\mathbf{k} = \begin{matrix} \\ \mathbf{U} \\ \mathbf{V} \end{matrix} \overset{\begin{matrix} \mathbf{U} & \mathbf{V} \end{matrix}}{\begin{bmatrix} \mathbf{k}^{uu} & \mathbf{k}^{uv} \\ \mathbf{k}^{vu} & \mathbf{k}^{vv} \end{bmatrix}} \tag{5.25}$$

where

$$\begin{aligned}
\mathbf{k}^{uu} &= \frac{ab}{4}\left[C_{11}\frac{\xi_k\xi_m}{a^2}\left(1+\frac{1}{3}\eta_k\eta_m\right)+C_{33}\frac{\eta_k\eta_m}{b^2}\left(1+\frac{1}{3}\xi_k\xi_m\right)\right],\\
\mathbf{k}^{uv} &= \frac{1}{4}\left[C_{12}\xi_k\eta_m + C_{33}\eta_k\xi_m\right],\\
\mathbf{k}^{vu} &= \left(\mathbf{k}^{uv}\right)^{\mathrm{T}},\\
\mathbf{k}^{vv} &= \frac{ab}{4}\left[C_{22}\frac{\eta_k\eta_m}{b^2}\left(1+\frac{1}{3}\xi_k\xi_m\right)+C_{33}\frac{\xi_k\xi_m}{a^2}\left(1+\frac{1}{3}\eta_k\eta_m\right)\right].
\end{aligned} \tag{5.26}$$

For the case of plane stress with $\nu = 1/3$ and $a = b$, the stiffness matrix is:

$$\mathbf{k} = \frac{\mu}{24}\begin{array}{c} \begin{matrix} U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \begin{bmatrix} 32 & -20 & -16 & 4 & 12 & 0 & -12 & 0 \\ -20 & 32 & 4 & -16 & 0 & -12 & 0 & 12 \\ -16 & 4 & 32 & -20 & -12 & 0 & 12 & 0 \\ 4 & -16 & -20 & 32 & 0 & 12 & 0 & -12 \\ 12 & 0 & -12 & 0 & 32 & 4 & -16 & -20 \\ 0 & -12 & 0 & 12 & 4 & 32 & -20 & -16 \\ -12 & 0 & 12 & 0 & -16 & -20 & 32 & 4 \\ 0 & 12 & 0 & -12 & -20 & -16 & 4 & 32 \end{bmatrix} \end{array}. \tag{5.27}$$

This can be compared with the stiffness matrix 4.50 derived from four triangles by static condensation:

$$\mathbf{k} = \frac{\mu}{24}\begin{array}{c} \begin{matrix} U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \begin{bmatrix} 33 & -21 & -15 & 3 & 12 & 0 & -12 & 0 \\ -21 & 33 & 3 & -15 & 0 & -12 & 0 & 12 \\ -15 & 3 & 33 & -21 & -12 & 0 & 12 & 0 \\ 3 & -15 & -21 & 33 & 0 & 12 & 0 & -12 \\ 12 & 0 & -12 & 0 & 33 & 3 & -15 & -21 \\ 0 & -12 & 0 & 12 & 3 & 33 & -21 & -15 \\ -12 & 0 & 12 & 0 & -15 & -21 & 33 & 3 \\ 0 & 12 & 0 & -12 & -21 & -15 & 3 & 33 \end{bmatrix} \end{array}. \tag{5.28}$$

For plane strain with $\nu = 1/3$ and $a = b$, Equation 5.25 gives

$$\mathbf{k} = \frac{\mu}{24}\begin{array}{c} \begin{matrix} U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \begin{bmatrix} 40 & & & & & & & \\ -28 & 40 & & & & & & \\ -20 & 8 & 40 & & & & & \\ 8 & -20 & -28 & 40 & & & & \\ 18 & -6 & -18 & 6 & 40 & & & \\ 6 & -18 & -6 & 18 & 8 & 40 & & \\ -18 & 6 & 18 & -6 & -20 & -28 & 40 & \\ -6 & 18 & 6 & -18 & -28 & -20 & 8 & 40 \end{bmatrix} \end{array}. \tag{5.29}$$

Equations 5.27 and 5.29 can be compared with the output of ANSYS for the element Plane42 excluding the extra DOFs. We will introduce the extra DOF later. Although this element is very instructive and valuable for historical reasons, ANSYS is phasing it out. I will continue to refer to it as a synonym for an analysis based on the developments of this section and the Wilson–Taylor element.

5.1.1 Stress Calculations

The stress is given by

$$\boldsymbol{\tau} = \mathbf{C}\boldsymbol{\varepsilon} = \mathbf{CAD} = \mathbf{BD}. \tag{5.30}$$

Since $\mathbf{A}$ is not constant for this element, the stress and strain vary within an element. At the centroid $\xi = 0$ and $\eta = 0$, for $a = b$,

$$\mathbf{A} = \frac{1}{2a}\begin{bmatrix} -1 & 1 & 1 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 & -1 & 1 & 1 & -1 \end{bmatrix}. \tag{5.31}$$

For plane stress with $\nu = 1/3$,

$$\mathbf{C} = \frac{3E}{8}\begin{bmatrix} 3 & 1 & 0 \\ 1 & 3 & 0 \\ 0 & 0 & 1 \end{bmatrix}. \tag{5.32}$$

Therefore, at the centroid, for plane stress with $\nu = 1/3$,

$$\mathbf{B} = \mathbf{CA} = \frac{3E}{16a}\begin{bmatrix} -3 & 3 & 3 & -3 & -1 & -1 & 1 & 1 \\ -1 & 1 & 1 & -1 & -3 & -3 & 3 & 3 \\ -1 & -1 & 1 & 1 & -1 & 1 & 1 & -1 \end{bmatrix}. \tag{5.33}$$

For node 3, $\xi = 1$ and $\eta = 1$. For $a = b$,

$$\mathbf{A} = \frac{1}{a}\begin{bmatrix} 0 & 0 & 1 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 & 1 & 0 \\ 0 & -1 & 1 & 0 & 0 & 0 & 1 & -1 \end{bmatrix}. \tag{5.34}$$

Therefore, at node 3, for plane stress with $\nu = 1/3$,

$$\mathbf{B} = \frac{3E}{8a}\begin{bmatrix} 0 & 0 & 3 & -3 & 0 & -1 & 1 & 0 \\ 0 & 0 & 1 & -1 & 0 & -3 & 3 & 0 \\ 0 & -1 & 1 & 0 & 0 & 0 & 1 & -1 \end{bmatrix}. \tag{5.35}$$

For the Plane42 element, by default, ANSYS provides the stress evaluated at the nodes in the element listing and the stress at the centroid in the Element Table (ETABLE). Stress at the integration points (IPs), which will be defined below, is available in the element listing and the nodal stress listing if ERESX = No, Solution > Load Step Options > Output Ctrls > Integration Point. The IP stresses are then copied to the nodal points, while the element table still has the stress at the centroid.

For the Plane182 element, by default, ANSYS provides the stress evaluated at the nodes in the element listing. Stress at the IP is available in the element listing and the nodal stress listing if ERESX = No; the IP stresses are then copied to the nodal points. There is no ETABLE for stress.

The components of stress are generally not continuous across element boundaries, so the value at a node may be different for all elements sharing that node. The stress at the node as reported by ANSYS is the average of values for all elements sharing the node.

5.1.2 Plane Stress Example: Pure Bending

The support conditions (Figure 5.2) are zero displacement of the origin and $v(a,0) = 0$. There is symmetry (S: $u = 0$, $\tau_{xy} = 0$) about the centerline ($x = a/2$) and antisymmetry (A: $u = 0$, $\tau_{yy} = 0$) about the x axis, so we need only analyze one-quarter of the

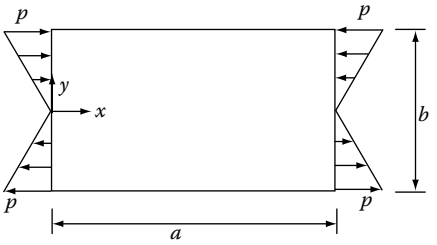


FIGURE 5.2 Pure bending.

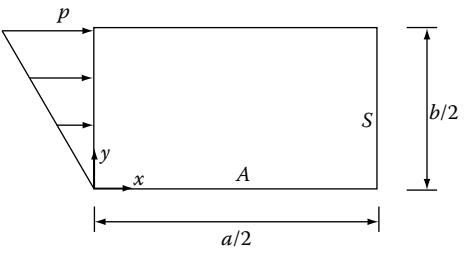


FIGURE 5.3 Bending with symmetry.

TABLE 5.1
Nodal Values

N	$v(1,0)$	$\tau_{xx}(1,1)$
1	0.44608	0.93627
2	0.48528	0.98985
4	0.49643	1.0039
8	0.49911	1.0041
16	0.49978	1.0026
32	0.49994	1.0014
64	0.49999	1.0007
Exact	0.5	1

region with zero displacement at the origin (Figure 5.3). The region can be divided into square elements by N divisions on an edge to produce N^2 elements and $(N + 1)^2$ nodes. For $a = b = 2$, $p = 1$, $E = 1$, $\nu = 0.3$, the solution is shown in Table 5.1.

In this case, the actual distribution of displacement is close to that of the shape functions so relatively few elements are needed for a close answer. Let us investigate the case $N = 2$, shown in Figure 5.4 in more detail. From 5.14, we see that the stress components will vary linearly in y when x is constant. The element output for element 2 is $\tau_{xx} = 0.51426$ at node 5. The element output for element 4 is $\tau_{xx} = 0.46632$ at node 5. The stress tensor is not continuous between elements and the data have to be smoothed in some fashion in order to obtain a reasonable estimate of the stress at the element boundary. ANSYS does that by averaging the values from all elements connected to a node. The reported stress at node 5 is therefore $\tau_{xx} = (0.51426 + 0.46632)/2 = 0.49029$.

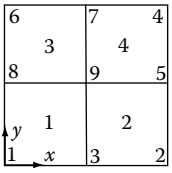


FIGURE 5.4 A 2 × 2 element layout.

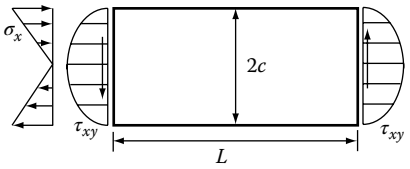


FIGURE 5.5 Plane strain bending with shear.

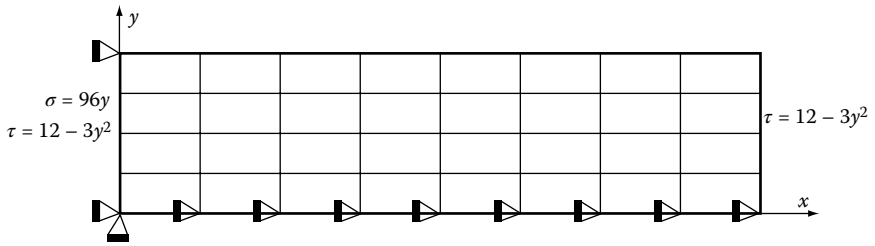


FIGURE 5.6 Bending with shear: element layout.

5.1.3 Plane Strain Example: Bending with Shear

The exact solution for the plane strain of a rectangular region loaded by parabolic shear stress on one end (Figure 5.5) is given in Section 2.1.12. We will now apply the finite element method (FEM) using the four-node element to obtain an approximate solution for the case $c = 2$, $L = 16$, $P = 32$, $E = 8192$, $\nu = 0.3$. Symmetry about the x axis can be used. The mesh chosen has N divisions along the y axis and M divisions along the x axis. A 4×8 mesh for the upper half is shown (Figure 5.6). The coordinate axes have been shifted for computational convenience.

The edge $y = 0$ has $u_x = 0$ by symmetry. To prevent rigid body motion, we will fix the origin and set $u_x = 0$ at the upper left corner ($x = 0$, $y = c$). The exact solution for the vertical displacement at the center of the right edge is

$$u_y = \frac{PL^3(1-\nu^2)}{2Ec^3}\left(1+\frac{c^2}{L^2}\frac{(4+\nu)}{2(1-\nu)}\right). \tag{5.36}$$

Table 5.2 shows the FEM solution obtained from ANSYS using the Plane42 element without extra DOF. From 5.36, the exact displacement is 0.9537 for $\nu = 0.3$ and 0.8037 for $\nu = 0.499$. Note the relatively poor solution for the Poisson ratio near 0.5 when the bulk modulus is very large compared to the shear modulus.

5.1.4 Plane Stress Example: Short Beam

The same plane stress problem that was analyzed in Section 4.4 using constant strain triangles will be analyzed by using the four-node rectangular elements. The strains, and therefore the stresses, vary linearly within each element in this case. The finite element layout is shown in Figure 5.7. There are N segments per side, N^2 rectangles,

TABLE 5.2
Displacements for Different Poisson Ratio

$N \times M$	ν	u_y	FEM/EXACT
4 × 8	0.3	0.847	0.888
8 × 32	0.3	0.940	0.986
16 × 64	0.3	0.950	0.996
4 × 8	0.499	0.264	0.328
8 × 32	0.499	0.529	0.658
16 × 64	0.499	0.703	0.875

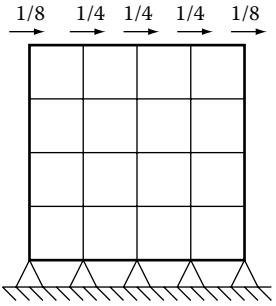


FIGURE 5.7 Short beam: element layout.

and $(N + 1)^2$ nodes. The nodes along the bottom have zero displacement and the nodes along the top have generalized forces as shown.

The displacement of the corner is listed in Table 5.3 for several mesh sizes, for $\nu = 1/3$. The stress distribution for the case of 2500 rectangles is presented in Figures 5.8 through 5.10. The element stress is calculated at the centroid of the element using 5.33.

TABLE 5.3
Convergence

Segments per Side	No. of Nodes	No. of Rectangles	x Displacement of the Corner
1	4	1	4.667
2	9	4	5.908
4	25	16	6.792
10	121	100	7.240
20	441	400	7.355
46	2,209	2,216	7.409
50	2,601	2,500	7.412
60	3,721	3,600	7.417
70	5,041	4,900	7.421
80	6,561	6,400	7.424
90	8,281	8,100	7.426
100	1,020	10,000	7.428

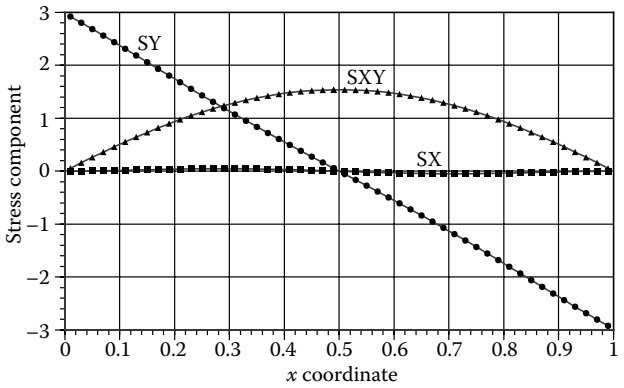


FIGURE 5.8 Short beam: stress at $y = 0.51$.

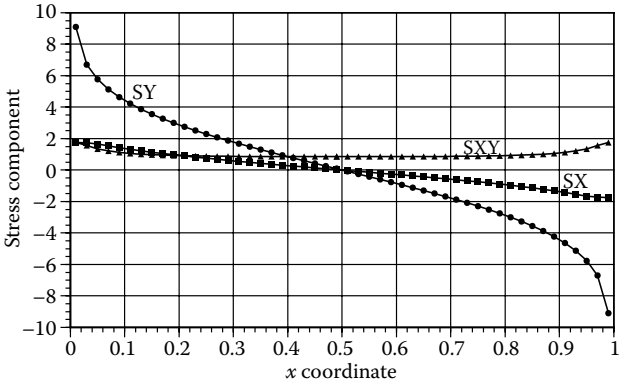


FIGURE 5.9 Short beam: stress at $y = 0.01$.

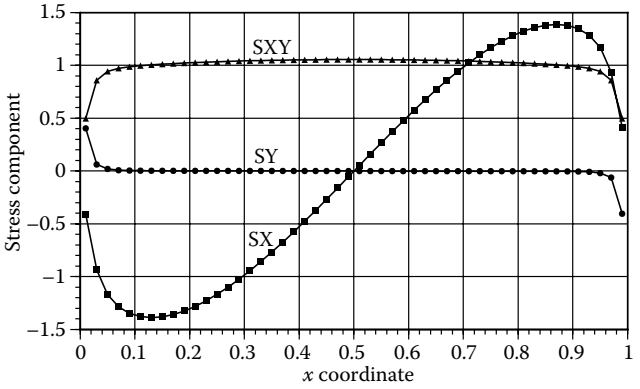


FIGURE 5.10 Short beam: stress at $y = 0.99$.

5.2 IMPROVEMENTS TO FOUR-NODE QUADRILATERAL

5.2.1 Wilson–Taylor Quadrilateral

For clarity of presentation and organization in the general discussion of the finite element formulation, we have assumed that the DOFs D_K are the values of the displacement at some node. However, the calculations made no use of that fact and the same equations are obtained even if the displacement parameters have no kinematical meaning. We will now consider an example of such a generalization.

The shape functions 5.6 are an incomplete quadratic form. Consequently, the strains and stresses are represented by incomplete linear forms. For example, $\varepsilon_{11} = c_2 + c_4 y$. The representation of the x dependence is not as complete as that for y. This does not give a uniform approximation in the general case. However, in order to obtain a complete linear form we would need at least six nodes, and there is no natural way to choose them. This dilemma can be circumvented by introducing nodeless parameters such as the following[1]:

$$\begin{aligned} u &= \sum_{i=1}^{4} N_i(\xi,\eta)U_i + \frac{a}{4}(\xi^2-1)d_1 + \frac{b}{4}(\eta^2-1)d_4, \\ v &= \sum_{i=1}^{4} N_i(\xi,\eta)V_i + \frac{b}{4}(\eta^2-1)d_2 + \frac{a}{4}(\xi^2-1)d_3. \end{aligned} \tag{5.37}$$

The new terms with displacement parameters d_i are zero at the corners of the rectangle so that the interpretation of U_i and V_i as x and y components of the nodal displacement is unchanged. The d_i values are not given any particular geometrical interpretation.

If we expand the representation for u into powers of x and y, we obtain the complete quadratic form:

$$u = c_1 + c_2 x + c_3 y + c_4 x^2 + c_5 xy + c_6 y^2. \tag{5.38}$$

Therefore, the strains and stresses will be represented by complete linear forms, for example,

$$\varepsilon_{11} = c_2 + c_4 x + c_5 y. \tag{5.39}$$

This will give a representation of stress that is not biased toward one particular coordinate. The matrix $\mathbf{D}$ of displacement parameters for an element is now

$$[U_1\ U_2\ U_3\ U_4\ V_1\ V_2\ V_3\ V_4\ d_1\ d_2\ d_3\ d_4] = [\mathbf{D}^{\mathrm{T}}, \mathbf{d}^{\mathrm{T}}]. \tag{5.40}$$

The strains are

$$\boldsymbol{\varepsilon} = \mathbf{AD} + \mathbf{Bd}, \tag{5.41}$$

where $\mathbf{A}$ is given by 5.20 for the four-node rectangle and

$$\mathbf{B} = \begin{bmatrix} \xi & 0 & 0 & 0 \\ 0 & \eta & 0 & 0 \\ 0 & 0 & \xi & \eta \end{bmatrix}. \tag{5.42}$$

The potential energy is therefore

$$\mathcal{P} = \frac{1}{2}[\mathbf{D}^{\mathrm{T}}\ \mathbf{d}^{\mathrm{T}}] \begin{bmatrix} \mathbf{k}_{11} & \mathbf{k}_{12} \\ \mathbf{k}_{21} & \mathbf{k}_{22} \end{bmatrix} \begin{bmatrix} \mathbf{D} \\ \mathbf{d} \end{bmatrix} - \mathbf{D}^{\mathrm{T}}\mathbf{F} \tag{5.43}$$

where $\mathbf{k}_{11}$ is the element stiffness matrix for the four-node rectangle and

$$\begin{aligned} \mathbf{k}_{12} = \mathbf{k}_{21}^{\mathrm{T}} &= \iint \mathbf{A}^{\mathrm{T}}\mathbf{CB}\,\mathrm{d}A, \\ \mathbf{k}_{22} &= \iint \mathbf{B}^{\mathrm{T}}\mathbf{CB}\,\mathrm{d}A. \end{aligned} \tag{5.44}$$

The finite element equations for the element are therefore

$$\begin{aligned} \mathbf{k}_{11}\mathbf{D} + \mathbf{k}_{12}\mathbf{d} &= \mathbf{F}, \\ \mathbf{k}_{21}\mathbf{D} + \mathbf{k}_{22}\mathbf{d} &= \mathbf{0}. \end{aligned} \tag{5.45}$$

The element stiffness matrix in 5.43 is a 12×12 matrix. In an assembly of elements, the parameters d_i are not shared with any other elements. We have four different parameters for each element. The four parameters d_i can therefore be eliminated for each element:

$$\mathbf{d} = -\mathbf{k}_{22}^{-1}\mathbf{k}_{21}\mathbf{D}, \tag{5.46}$$

and

$$\mathbf{k} = \mathbf{k}_{11} - \mathbf{k}_{12}\ \mathbf{k}_{22}^{-1}\ \mathbf{k}_{21}. \tag{5.47}$$

This process is called static condensation. The result 5.47 is an 8 × 8 element stiffness matrix that may be used in place of the four-node serendipity element. This new matrix will give improved displacements and stresses for a given number of rectangles. The results for the short beam example are shown in Figure 5.11. The results are significantly improved for a small number of elements, and the result appears to converge to the same result as for the four-node rectangles.

For $a = b$ and $\nu = 1/3$, the first row is

$$\begin{array}{lcccccccccc} & U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 & \\ [k_{1i}] = \dfrac{\mu}{36}\Big[& 54 & -36 & -36 & 18 & 27 & 9 & -27 & -9 \Big] & \text{plane strain.} \\ [k_{1i}] = \dfrac{\mu}{36}\Big[& 44 & -26 & -28 & 10 & 18 & 0 & -18 & 0 \Big] & \text{plane stress.} \end{array} \tag{5.48}$$

This agrees with ANSYS element Plane42 with extra DOFs.

If we investigate the displacement field for an assembly of nonrectangular elements, we find that representation 5.37 does not generate a continuous displacement field. Because of the ξ^2 and η^2 terms, the displacement is not varying linearly along a side unless $d_i = 0$. This violates the assumptions in the derivation of the finite element equations from the potential energy.

Elements that do not generate continuous displacement fields when they are assembled are called *incompatible* or *nonconforming*. We will see later that this particular nonconforming element is acceptable as long as the element is rectangular, or the corresponding isoparametric element is a parallelogram. A modified integration scheme has been invented that removes the restriction to parallelograms.[2]

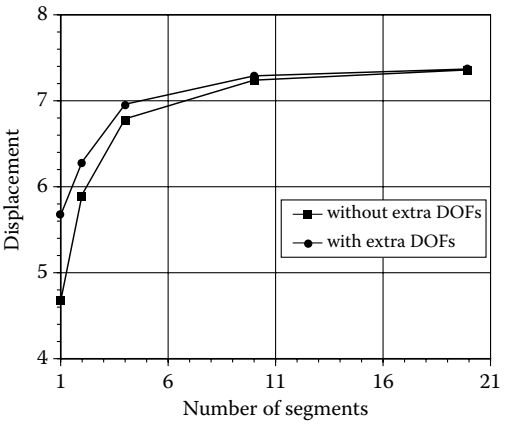


FIGURE 5.11 Comparison with extra DOF.

It is presented in Section 6.3.1. The Wilson–Taylor element is a special case of the following enhanced strain element.

5.2.2 Enhanced Strain Formulation

A potential energy function for the Wilson–Taylor element can be established by regarding the extra strain resulting from the nodeless parameters as an independent variable that is not derived from the displacements.[3] Let us consider the modified potential energy function

$$\tilde{\mathcal{P}}(\mathbf{u},\boldsymbol{\tau},\tilde{\boldsymbol{\varepsilon}}) = \int_{\mathcal{V}} \left(\mathcal{A}(\boldsymbol{\varepsilon}+\tilde{\boldsymbol{\varepsilon}}) - \tau_{ij}\tilde{\varepsilon}_{ij} \right) \mathrm{d}V - \int_{\mathcal{V}} b_i u_i \, \mathrm{d}V - \int_{S_T} T_i^0 u_i \, \mathrm{d}A, \tag{5.49}$$

where $\boldsymbol{\varepsilon}$ is derived from $\mathbf{u}$ by the strain–displacement relations and $\tilde{\boldsymbol{\varepsilon}}$ is a parameter called the enhanced strain tensor. Continuity is required for the displacement $\mathbf{u}$ but not for the stress $\boldsymbol{\tau}$ nor the enhanced strain $\tilde{\boldsymbol{\varepsilon}}$. If $\tilde{\boldsymbol{\varepsilon}} = 0$, 5.49 reduces to the potential energy formula. For linear elasticity,

$$\mathcal{A}(\boldsymbol{\varepsilon}+\tilde{\boldsymbol{\varepsilon}}) = \frac{1}{2} c_{ijkm} \left(\varepsilon_{ij} + \tilde{\varepsilon}_{ij} \right) \left(\varepsilon_{km} + \tilde{\varepsilon}_{km} \right), \tag{5.50}$$

with $c_{ijkm} = c_{jikm} = c_{ijmk} = c_{kmij}$. The derivative of $\mathcal{P}$ is

$$\begin{aligned} \delta\mathcal{P} = & \int_{\mathcal{V}} \left[c_{ijkm} \left(\varepsilon_{km} + \tilde{\varepsilon}_{km} \right) - \tau_{ij} \right] \delta\tilde{\varepsilon}_{ij} \, \mathrm{d}V - \int_{\mathcal{V}} \tilde{\varepsilon}_{ij} \, \delta\tau_{ij} \, \mathrm{d}V \\ & + \int_{\mathcal{V}} \left[c_{ijkm} \left(\varepsilon_{km} + \tilde{\varepsilon}_{km} \right) \right] \delta\varepsilon_{ij} \, \mathrm{d}V - \int_{\mathcal{V}} b_i \, \delta u_i \, \mathrm{d}V - \int_{S_T} T_i^0 \, \delta u_i \, \mathrm{d}A. \end{aligned} \tag{5.51}$$

Integrating by parts, subject to the condition that $\mathbf{u}$ is as continuous function and $\delta\mathbf{u} = 0$ on S_u,

$$\begin{aligned} D\mathcal{P} = & \int_{\mathcal{V}} \left[c_{ijkm} \left(\varepsilon_{km} + \tilde{\varepsilon}_{km} \right) - \tau_{ij} \right] \delta\tilde{\varepsilon}_{ij} \, \mathrm{d}V - \int_{\mathcal{V}} \tilde{\varepsilon}_{ij} \delta\tau_{ij} \, \mathrm{d}V \\ & - \int_{\mathcal{V}} \left[\left(c_{ijkm} \left(\varepsilon_{km} + \tilde{\varepsilon}_{km} \right) \right)_{,j} + b_i \right] \delta u_i \, \mathrm{d}V \\ & + \int_{S_T} \left[n_j c_{ijkm} \left(\varepsilon_{km} + \tilde{\varepsilon}_{km} \right) - T_i^0 \right] \delta u_i \, \mathrm{d}A. \end{aligned} \tag{5.52}$$

Thus, $D\mathcal{P} = 0$ is equivalent to

$$\begin{aligned} &\tilde{\varepsilon}_{ij} = 0, \\ &\tau_{ij} = c_{ijkm}\left(\varepsilon_{km} + \tilde{\varepsilon}_{km}\right), \\ &\tau_{ij,j} + b_i = 0, \end{aligned} \tag{5.53}$$

in $\mathcal{V}$, and

$$n_j\,\tau_{ij} = T_i^0 \tag{5.54}$$

on $\mathcal{S}_T$. These are the equations of linear elasticity.

We can now apply FEM to formulate a solution procedure. In matrix form, using 5.51, $D\mathcal{P} = 0$ gives

$$\begin{aligned} &\int_{\mathcal{V}} \delta\tilde{\boldsymbol{\varepsilon}}^{\mathrm{T}}\left[\mathbf{C}\boldsymbol{\varepsilon} + \mathbf{C}\tilde{\boldsymbol{\varepsilon}} - \boldsymbol{\tau}\right]\mathrm{d}V = 0, \\ &\int_{\mathcal{V}} \delta\boldsymbol{\tau}^{\mathrm{T}}\tilde{\boldsymbol{\varepsilon}}\,\mathrm{d}V = 0, \\ &\int_{\mathcal{V}} \delta\boldsymbol{\varepsilon}^{\mathrm{T}}\left[\mathbf{C}\boldsymbol{\varepsilon} + \mathbf{C}\tilde{\boldsymbol{\varepsilon}}\right]\mathrm{d}V = \int_{\mathcal{V}} \delta\mathbf{u}^{\mathrm{T}}\mathbf{b}\,\mathrm{d}V + \int_{\mathcal{S}_T} \delta\mathbf{u}^{\mathrm{T}}\mathbf{T}\,\mathrm{d}A. \end{aligned} \tag{5.55}$$

We introduce shape functions for the three fields:

$$\mathbf{u} = \mathbf{N}\mathbf{D}, \quad \boldsymbol{\varepsilon} = \mathbf{A}\mathbf{D}, \quad \boldsymbol{\tau} = \mathbf{S}\boldsymbol{\sigma}, \quad \tilde{\boldsymbol{\varepsilon}} = \mathbf{G}\mathbf{d}. \tag{5.56}$$

Both $\mathbf{d}$ and $\boldsymbol{\sigma}$ are nodeless parameters for each element and are not shared by other elements. Matrix $\mathbf{A}$ is determined by $\mathbf{N}$ through the strain–displacement relations. Substituting into 5.55 and merging the element equations, we find from the second equation that

$$\int_{\mathcal{V}} \mathbf{S}^{\mathrm{T}}\mathbf{G}\,\mathrm{d}V\mathbf{d} = 0. \tag{5.57}$$

Therefore, we will require that the shape functions are such that

$$\int_{\mathcal{V}} \mathbf{S}^{\mathrm{T}}\mathbf{G}\,\mathrm{d}V = 0. \tag{5.58}$$

The first and third equations in 5.55 then reduce to

$$\mathbf{k}_{11}\mathbf{D} + \mathbf{k}_{12}\mathbf{d} = \mathbf{F}, \tag{5.59}$$

and

$$\mathbf{k}_{21}\mathbf{D} + \mathbf{k}_{22}\mathbf{d} = 0, \tag{5.60}$$

where

$$\begin{aligned}
\mathbf{k}_{11} &= \int_{\mathcal{V}} \mathbf{A}^{\mathrm{T}}\mathbf{C}\mathbf{A}\,\mathrm{d}V, \\
\mathbf{k}_{12} = \mathbf{k}_{21}^{\mathrm{T}} &= \int_{\mathcal{V}} \mathbf{G}^{\mathrm{T}}\mathbf{C}\mathbf{A}\,\mathrm{d}V, \\
\mathbf{k}_{22} &= \int_{\mathcal{V}} \mathbf{G}^{\mathrm{T}}\mathbf{C}\mathbf{G}\,\mathrm{d}V.
\end{aligned} \tag{5.61}$$

The force matrix is the same as for the pure displacement formulation and is not affected by the introduction of the enhanced strain. Since the parameters $\mathbf{d}$ are not shared by elements, they can be eliminated at the element level by static condensation:

$$\mathbf{d} = -\mathbf{k}_{22}^{-1}\,\mathbf{k}_{21}\mathbf{D}, \tag{5.62}$$

to obtain the element stiffness matrix

$$\mathbf{k} = \mathbf{k}_{11} - \mathbf{k}_{12}\mathbf{k}_{22}^{-1}\mathbf{k}_{21}. \tag{5.63}$$

Wilson–Taylor Element. Let us now treat the plane problem using the four-node element with nodeless parameters and treat the nodeless parameters as generating the enhanced strain:

$$\mathbf{G} = \begin{bmatrix} \xi & 0 & 0 & 0 \\ 0 & \eta & 0 & 0 \\ 0 & 0 & \xi & \eta \end{bmatrix}. \tag{5.64}$$

In order to satisfy 5.58, we can chose

$$\mathbf{S} = \begin{bmatrix} 1 & 0 & 0 & \eta & 0 \\ 0 & 1 & 0 & 0 & \xi \\ 0 & 0 & 1 & 0 & 0 \end{bmatrix}. \tag{5.65}$$

For plane strain with $\nu = 1/3$,

$$\mathbf{C} = \mu \begin{bmatrix} 4 & 2 & 0 \\ 2 & 4 & 0 \\ 0 & 0 & 1 \end{bmatrix}. \tag{5.66}$$

For a square element,

$$\mathbf{k}_{22} = \frac{ab\mu}{3}\begin{bmatrix} 4 & 0 & 0 & 0 \\ 0 & 4 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}, \tag{5.67}$$

and

$$\mathbf{k}_{21} = \frac{\mu a}{6}\begin{bmatrix} 0 & 2 & 0 & -2 & 0 & 2 & 0 & -2 \\ 2 & 0 & -2 & 0 & 2 & 0 & -2 & 0 \\ 1 & 0 & -1 & 0 & 1 & 0 & -1 & 0 \\ 0 & 1 & 0 & -1 & 0 & 1 & 0 & -1 \end{bmatrix}. \tag{5.68}$$

Therefore,

$$\mathbf{k}_{12}\mathbf{k}_{22}^{-1}\mathbf{k}_{21} = \frac{\mu}{6}\begin{array}{c} \begin{matrix} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 \end{matrix} \\ \begin{bmatrix} 1 & & & & & & & \\ 0 & 1 & & & & & & \\ -1 & 0 & 1 & & & & & \\ 0 & -1 & 0 & 1 & & & & \\ 1 & 0 & -1 & 0 & 1 & & & \\ 0 & 1 & 0 & -1 & 0 & 1 & & \\ -1 & 0 & 1 & 0 & -1 & 0 & 1 & \\ 0 & -1 & 0 & 1 & 0 & -1 & 0 & 1 \end{bmatrix} \end{array}. \tag{5.69}$$

Combined with 5.27, we find that the element stiffness matrix for plane strain is

$$\mathbf{k} = \frac{\mu}{24}\begin{array}{c} \begin{matrix} U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \begin{bmatrix} 36 & & & & & & & \\ -24 & 36 & & & & & & \\ -24 & 12 & 36 & & & & & \\ 12 & -24 & -24 & 36 & & & & \\ 18 & -6 & -18 & 6 & 36 & & & \\ 6 & -18 & -6 & 8 & 12 & 36 & & \\ -18 & 6 & 18 & -6 & -24 & -24 & 36 & \\ -6 & 18 & 6 & -18 & -24 & -24 & 12 & 36 \end{bmatrix} \end{array}. \tag{5.70}$$

This is the same as for ANSYS Plane42 element with extra DOF and the ANSYS Plane182 element with enhanced strain.

5.2.3 Approximate Volumetric Strains

We see from Table 5.2 that the solution when the Poisson ration is near 0.5 is poor for any mesh. The basic serendipity element is apparently giving too much weight to the volume change relative to the distortion because of the large bulk modulus, that is, the element is too stiff in volume change. The performance can be improved by using a lower-order integration on the volumetric terms. We will explore that option in this and the next two sections.

For the plane problem $\gamma_{xz} = \gamma_{yz} = 0$ and

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \sigma_z \end{bmatrix} = \mathbf{C} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \varepsilon_z \end{bmatrix} \tag{5.71}$$

where

$$\mathbf{C} = \begin{bmatrix} 2\mu+\lambda & \lambda & 0 & \lambda \\ \lambda & 2\mu+\lambda & 0 & \lambda \\ 0 & 0 & \mu & 0 \\ \lambda & \lambda & 0 & 2\mu+\lambda \end{bmatrix} \tag{5.72}$$

and

$$\varepsilon_z = \begin{cases} 0 & \text{for plane strain.} \\ -\dfrac{\nu}{1-\nu}\left(\varepsilon_x + \varepsilon_y\right) & \text{for plane stress.} \end{cases} \tag{5.73}$$

The total strains can be expressed in terms of the deviatoric strains and the volumetric strain

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \varepsilon_z \end{bmatrix} = \begin{bmatrix} e_x \\ e_y \\ \gamma_{xy} \\ e_z \end{bmatrix} + \frac{1}{3}\begin{bmatrix} \varepsilon_v \\ \varepsilon_v \\ 0 \\ \varepsilon_v \end{bmatrix}. \tag{5.74}$$

From 5.19,

$$\varepsilon_x = \mathbf{A}_1\mathbf{U}, \quad \varepsilon_y = \mathbf{A}_2\mathbf{V}, \quad \gamma_{xy} = \mathbf{A}_2\mathbf{U} + \mathbf{A}_1\mathbf{V}. \tag{5.75}$$

For plane strain,

$$\varepsilon_v = \varepsilon_x + \varepsilon_y = \begin{bmatrix} \mathbf{A}_1 & \mathbf{A}_2 \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}. \tag{5.76}$$

Therefore, for plane strain,

$$\begin{bmatrix} e_x \\ e_y \\ \gamma_{xy} \\ e_z \end{bmatrix} = \begin{bmatrix} \mathbf{A}_1 & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_2 \\ \mathbf{A}_2 & \mathbf{A}_1 \\ \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix} - \frac{1}{3} \begin{bmatrix} \mathbf{A}_1 & \mathbf{A}_2 \\ \mathbf{A}_1 & \mathbf{A}_2 \\ \mathbf{0} & \mathbf{0} \\ \mathbf{A}_1 & \mathbf{A}_2 \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix} = \begin{bmatrix} \frac{2}{3}\mathbf{A} & -\frac{1}{3}\mathbf{B} \\ -\frac{1}{3}\mathbf{A} & \frac{2}{3}\mathbf{B} \\ \mathbf{B} & \mathbf{A} \\ -\frac{1}{3}\mathbf{A} & -\frac{1}{3}\mathbf{B} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}. \tag{5.77}$$

Define

$$\mathbf{A}^{(\mathrm{dil})} = \frac{1}{3} \begin{bmatrix} \mathbf{A}_1 & \mathbf{A}_2 \\ \mathbf{A}_1 & \mathbf{A}_2 \\ \mathbf{0} & \mathbf{0} \\ \mathbf{A}_1 & \mathbf{A}_2 \end{bmatrix} \tag{5.78}$$

and

$$\mathbf{A}^{(\mathrm{dev})} = \begin{bmatrix} \frac{2}{3}\mathbf{A}_1 & -\frac{1}{3}\mathbf{A}_2 \\ -\frac{1}{3}\mathbf{A}_1 & \frac{2}{3}\mathbf{A}_2 \\ \mathbf{A}_2 & \mathbf{A}_1 \\ -\frac{1}{3}\mathbf{A}_1 & -\frac{1}{3}\mathbf{A}_2 \end{bmatrix}. \tag{5.79}$$

Thus,

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \varepsilon_z \end{bmatrix} = \mathbf{A}^{(\mathrm{dev})} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix} + \mathbf{A}^{(\mathrm{dil})} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}. \tag{5.80}$$

We now approximate the dilatational term by evaluating matrix $\mathbf{A}^{(\mathrm{dil})}$ at the centroid ($\xi = \eta = 0$) of the element. This procedure is known as the $\bar{B}$ method.[4] Then,

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \varepsilon_z \end{bmatrix} = \bar{\mathbf{A}} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix} \tag{5.81}$$

where

$$\bar{\mathbf{A}} = \begin{bmatrix} \frac{2}{3}\mathbf{A}_1 + \frac{1}{3}\bar{\mathbf{A}}_1 & -\frac{1}{3}\mathbf{A}_2 + \frac{1}{3}\bar{\mathbf{A}}_2 \\ -\frac{1}{3}\mathbf{A}_1 + \frac{1}{3}\bar{\mathbf{A}}_1 & \frac{2}{3}\mathbf{A}_2 + \frac{1}{3}\bar{\mathbf{A}}_2 \\ \mathbf{A}_2 & \mathbf{A}_1 \\ -\frac{1}{3}\mathbf{A}_1 + \frac{1}{3}\bar{\mathbf{A}}_1 & -\frac{1}{3}\mathbf{A}_2 + \frac{1}{3}\bar{\mathbf{A}}_2 \end{bmatrix} \tag{5.82}$$

and

$$\bar{\mathbf{A}}_1 = \frac{1}{2a}\lfloor \xi_k \rfloor, \quad \bar{\mathbf{A}}_2 = \frac{1}{2b}\lfloor \eta_k \rfloor. \tag{5.83}$$

The element stiffness matrix for plane strain, therefore, is

$$\mathbf{k} = \int \bar{\mathbf{A}}^{\mathrm{T}} \mathbf{C} \bar{\mathbf{A}} \, \mathrm{d}a = \begin{bmatrix} \mathbf{k}_{11} & \mathbf{k}_{12} \\ \mathbf{k}_{21} & \mathbf{k}_{22} \end{bmatrix}. \tag{5.84}$$

Using 5.24,

$$\mathbf{k}_{11} = \frac{ab}{4} \begin{bmatrix} \frac{2\mu}{3a^2}\left[2\xi_k\xi_m \left(1 + \frac{1}{3}\eta_k\eta_m \right) + \xi_k\xi_m \right] \\ + \frac{\lambda}{a^2}\left[\xi_k\xi_m \right] \\ + \frac{\mu}{b^2}\left[\eta_k\eta_m \left(1 + \frac{1}{3}\xi_k\xi_m \right) \right] \end{bmatrix}. \tag{5.85}$$

For plane strain with $a = b$ and $\nu = 1/3$,

$$\mathbf{k}_{11} = \frac{\mu}{36} \begin{bmatrix} 52 & -34 & -38 & 20 \\ -34 & 52 & 20 & -38 \\ -38 & 20 & 52 & -34 \\ 20 & -38 & -34 & 52 \end{bmatrix}. \tag{5.86}$$

This is similar to the ANSYS Plane182 element with full integration ($K_1 = 1$) for plane strain with $a = b$ and $\nu = 1/3$, which has as the first row $\begin{pmatrix} 51 & -33 & -39 & 21 \end{pmatrix}$.

5.2.4 Reduced Integration on the κ Term

In this section, we separate the shear modulus from the bulk modulus $\kappa = \lambda + 2\mu/3$. The elasticity matrix for plane strain 5.22, therefore, takes the form

$$\mathbf{C} = \mathbf{C}_1 + \mathbf{C}_2 \tag{5.87}$$

with

$$\mathbf{C}_1 = \frac{\mu}{3}\begin{bmatrix} 4 & -2 & 0 \\ -2 & 4 & 0 \\ 0 & 0 & 3 \end{bmatrix} \tag{5.88}$$

and

$$\mathbf{C}_2 = \kappa\begin{bmatrix} 1 & 1 & 0 \\ 1 & 1 & 0 \\ 0 & 0 & 0 \end{bmatrix}. \tag{5.89}$$

Formula 5.21 for the element stiffness matrix becomes

$$\mathbf{k} = \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1} \mathbf{A}^T\mathbf{C}_1\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta + \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1} \mathbf{A}^T\mathbf{C}_2\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta. \tag{5.90}$$

Let us approximate the second term by treating the integrand as constant, equal to its value at the centroid ($\xi = 0, \eta = 0$) of the element (i.e., use one point numerical integration):

$$\mathbf{k} = \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1} \mathbf{A}^{\mathrm{T}}\mathbf{C}_1\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta + ab\mathbf{A}(0,0)^{\mathrm{T}}\mathbf{C}_2\mathbf{A}(0,0). \tag{5.91}$$

Using 5.20 and 5.24, we find for this case

$$\begin{aligned}
\mathbf{k}^{uu} &= \frac{ab}{4}\left[\frac{4\mu}{3}\frac{\xi_k\xi_m}{a^2}\left(1+\frac{1}{3}\eta_k\eta_m\right)+\mu\frac{\eta_k\eta_m}{b^2}\left(1+\frac{1}{3}\xi_k\xi_m\right)\right]+\kappa\left[\frac{b}{4a}\xi_k\xi_m\right], \\
\mathbf{k}^{uv} &= \frac{1}{4}\left[-\frac{2\mu}{3}\xi_k\eta_m+\mu\eta_k\xi_m\right]+\frac{\kappa}{4}\left[\eta_k\eta_m\right], \\
\mathbf{k}^{vu} &= (\mathbf{k}^{uv})^{\mathrm{T}}, \\
\mathbf{k}^{vv} &= \frac{ab}{4}\left[\frac{4\mu}{3}\frac{\eta_k\eta_m}{b^2}\left(1+\frac{1}{3}\xi_k\xi_m\right)+\mu\frac{\xi_k\xi_m}{a^2}\left(1+\frac{1}{3}\eta_k\eta_m\right)\right]+\kappa\left[\frac{b}{4a}\eta_k\eta_m\right].
\end{aligned} \tag{5.92}$$

For $a = b$ and $\nu = 1/3$,

$$[k_{1i}] = \frac{\mu}{36}\begin{matrix}\begin{bmatrix}52 & -34 & -38 & 20 & 27 & 9 & -27 & -9\end{bmatrix}\end{matrix} \quad \begin{matrix}U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4\end{matrix} \tag{5.93}$$

in agreement with 5.86.

5.2.5 Reduced Integration on the λ Term

In this section, we separate the dependence on μ and $\lambda = 2\mu\nu/(1 - 2\nu)$ since λ is large for ν near 0.5. For plane strain,

$$\mathbf{C} = \mathbf{C}_1 + \mathbf{C}_2, \tag{5.94}$$

where

$$\mathbf{C}_1 = \mu\begin{bmatrix}2 & 0 & 0\\ 0 & 2 & 0\\ 0 & 0 & 1\end{bmatrix} \tag{5.95}$$

and

$$\mathbf{C}_2 = \lambda\begin{bmatrix}1 & 1 & 0\\ 1 & 1 & 0\\ 0 & 0 & 0\end{bmatrix}. \tag{5.96}$$

Formula 5.21 for the element stiffness matrix then becomes

$$\mathbf{k} = \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1}\mathbf{A}^{\mathrm{T}}\mathbf{C}_1\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta + \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1}\mathbf{A}^{\mathrm{T}}\mathbf{C}_2\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta. \tag{5.97}$$

Let us approximate the second term by treating the integrand as constant, equal to its value at the centroid ($\xi = 0, \eta = 0$) of the element (i.e., use one-point numerical integration):

$$\mathbf{k} = \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1}\mathbf{A}^{\mathrm{T}}\mathbf{C}_1\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta + ab\mathbf{A}(0,0)^{\mathrm{T}}\mathbf{C}_2\mathbf{A}(0,0). \tag{5.98}$$

Using 5.20, 5.95, and 5.96,

$$\mathbf{k} = \frac{ab}{4}\int_{\xi=-1}^{1}\int_{\eta=-1}^{1}\mu\begin{bmatrix} 2\mathbf{A}_1^{\mathrm{T}}\mathbf{A}_1 + \mathbf{A}_2^{\mathrm{T}}\mathbf{A}_2 & \mathbf{A}_2^{\mathrm{T}}\mathbf{A}_1 \\ \mathbf{A}_1^{\mathrm{T}}\mathbf{A}_2 & 2\mathbf{A}_2^{\mathrm{T}}\mathbf{A}_2 + \mathbf{A}_1^{\mathrm{T}}\mathbf{A}_1 \end{bmatrix} \mathrm{d}\xi\,\mathrm{d}\eta + ab\lambda\begin{bmatrix} \mathbf{A}_1(0)^{\mathrm{T}}\mathbf{A}_1(0) & \mathbf{A}_1(0)^{\mathrm{T}}\mathbf{A}_2(0) \\ \mathbf{A}_2(\mathbf{0})^{\mathrm{T}}\mathbf{A}_1(0) & \mathbf{A}_2(0)^{\mathrm{T}}\mathbf{A}_2(0) \end{bmatrix}. \tag{5.99}$$

For a square element with $\nu = 1/3$, $\lambda = 2\mu$, for plane strain,

$$\mathbf{k} = \frac{\mu}{8}\begin{array}{c} \begin{matrix} U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \begin{bmatrix} 12 & -8 & -8 & 4 & 6 & 2 & -6 & -2 \\ -8 & 12 & 4 & -8 & -2 & -6 & 2 & 6 \\ -8 & 4 & 12 & -8 & -6 & -2 & 6 & 2 \\ 4 & -8 & -8 & 12 & 2 & 6 & -2 & -6 \\ 6 & -2 & -6 & 2 & 12 & 4 & -8 & -8 \\ 2 & -6 & -2 & 6 & 4 & 12 & -8 & -8 \\ -6 & 2 & 6 & -2 & -8 & -8 & 12 & 4 \\ -2 & 6 & 2 & -6 & -8 & -8 & 4 & 12 \end{bmatrix} \end{array}. \tag{5.100}$$

This matrix agrees with the enhanced strain matrix 5.70.

5.2.6 Uniform Reduced Integration

Since the reduced integration of the λ terms results in such significant improvement, especially for a Poisson ratio near 0.5, one is tempted to try a reduced integration on both integrals. In such a case, formula 5.21 for the element stiffness matrix simplifies to

$$\mathbf{k} = ab\mathbf{A}(0,0)^{\mathrm{T}}\mathbf{C}\mathbf{A}(0,0) = \begin{array}{c} \begin{matrix} \mathbf{U} & \mathbf{V} \end{matrix} \\ \begin{bmatrix} \mathbf{k}^{uu} & \mathbf{k}^{uv} \\ \mathbf{k}^{vu} & \mathbf{k}^{vv} \end{bmatrix} \end{array}. \tag{5.101}$$

From 5.14, 5.20, and 5.22, for an isotropic material,

$$\begin{aligned} k_{km}^{uu} &= \frac{1}{4}\left[C_{11}\frac{b}{a}\xi_k\xi_m + C_{33}\frac{a}{b}\eta_k\eta_m\right], \\ k_{km}^{uv} &= \frac{1}{4}\left[C_{12}\xi_k\eta_m + C_{33}\eta_k\xi_m\right], \\ k_{km}^{vu} &= k_{mk}^{uv}, \\ k_{km}^{vv} &= \frac{1}{4}\left[C_{22}\frac{a}{b}\eta_k\eta_m + C_{33}\frac{b}{a}\xi_k\xi_m\right]. \end{aligned} \tag{5.102}$$

For a square element with $\nu = 1/3$, $\lambda = 2\mu$, for plane stress,

$$\mathbf{k} = \frac{\mu}{2}\begin{array}{c} \begin{matrix} U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \begin{bmatrix} 2 & -1 & -2 & 1 & 1 & 0 & -1 & 0 \\ -1 & 2 & 1 & -2 & 0 & -1 & 0 & 1 \\ -2 & 1 & 2 & -1 & -1 & 0 & 1 & 0 \\ 1 & -2 & -1 & 2 & 0 & 1 & 0 & -1 \\ 1 & 0 & -1 & 0 & 2 & 1 & -2 & -1 \\ 0 & -1 & 0 & 1 & 1 & 2 & -1 & -2 \\ -1 & 0 & 1 & 0 & -2 & -1 & 2 & 1 \\ 0 & 1 & 0 & -1 & -1 & -2 & 1 & 2 \end{bmatrix} \end{array}. \tag{5.103}$$

For plane strain,

$$\mathbf{k} = \frac{\mu}{4}\begin{bmatrix} 5 & -3 & -5 & 3 & 3 & 1 & -3 & -1 \\ -3 & 5 & 3 & -5 & -1 & -3 & 1 & 3 \\ -5 & 3 & 5 & -3 & -3 & -1 & 3 & 1 \\ 3 & -5 & -3 & 5 & 1 & 3 & -1 & -3 \\ 3 & -1 & -3 & 1 & 5 & 3 & -5 & -3 \\ 1 & -3 & -1 & 3 & 3 & 5 & -3 & -5 \\ -3 & 1 & 3 & -1 & -5 & -3 & 5 & 3 \\ -1 & 3 & 1 & -3 & -3 & -5 & 3 & 5 \end{bmatrix}. \tag{5.104}$$

A problem arises. The stiffness matrix contains other modes in addition to those for rigid body motion. There are five zero eigenvalues of matrix 5.104. Three correspond to the rigid body modes. The other two eigenvectors can be expressed by

$$\begin{array}{l} \begin{matrix} \quad U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix} \\ \mathbf{D}^{\mathrm{T}} = \begin{bmatrix} 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0 \end{bmatrix}, \\ \mathbf{D}^{\mathrm{T}} = \begin{bmatrix} 0 & 0 & 0 & 0 & 1 & -1 & 1 & -1 \end{bmatrix}. \end{array} \tag{5.105}$$

These are the so-called hourglass modes (Figure 5.12). If the element is restrained only against rigid motion, the determinant is still zero because of the hourglass modes.

This feature can be remedied by associating a stiffness with the hourglass modes.[5] The hourglass modes are $u = \xi\eta$ and $v = \xi\eta$. A similar exact solution of the plane equations is

$$u = \frac{\sigma}{c}xy, \quad v = \frac{\sigma}{2c}\left(\frac{b^2}{4} - x^2 - \alpha y^2\right), \quad \tau_{11} = \sigma y, \quad \tau_{22} = 0, \quad \tau_{12} = 0, \tag{5.106}$$

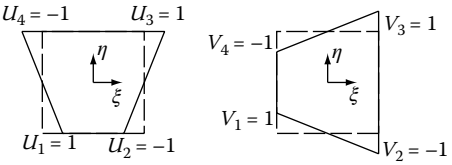


FIGURE 5.12 Hourglass modes.

where $c = E$ and $\alpha = \nu$ for plane stress, $c = 4\mu(\lambda + \mu)/(\lambda + 2\mu)$ and $\alpha = \nu/(1 - \nu)$ for plane strain. The nodal displacements are

$$U_n = -\frac{\sigma ab}{4c}(-1)^n, \tag{5.107}$$

which matches the hourglass mode. Since

$$\sum_{n=1}^{4}(-1)^n U_n = -\frac{\sigma ab}{c}, \tag{5.108}$$

we find

$$\sigma = -\frac{c}{ab}\sum_{n=1}^{4}(-1)^n U_n. \tag{5.109}$$

The nodal forces corresponding to this surface loading from 5.106 and 5.109 are calculated from the virtual work formula for each component of nodal displacement:

$$\mathbf{f} = \iint \mathbf{A}^{\mathrm{T}}\boldsymbol{\tau}\,\mathrm{d}x\,\mathrm{d}y = \iint \mathbf{A}^{\mathrm{T}}\begin{bmatrix}\sigma y\\ 0\\ 0\end{bmatrix}\mathrm{d}x\,\mathrm{d}y \tag{5.110}$$

where $\mathbf{A}$ is given by 5.20. Therefore,

$$\mathbf{f} = \frac{cb}{12a}\begin{bmatrix} 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0\\ -1 & 1 & -1 & 1 & 0 & 0 & 0 & 0\\ 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0\\ -1 & 1 & -1 & 1 & 0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{bmatrix} U_1\\ U_2\\ U_3\\ U_4\\ V_1\\ V_2\\ V_3\\ V_4 \end{bmatrix} \tag{5.111}$$

for the first hourglass mode. For the second mode,

$$\mathbf{f} = \frac{cb}{12a}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & -1 & 1 & -1 \\ 0 & 0 & 0 & 0 & -1 & 1 & -1 & 1 \\ 0 & 0 & 0 & 0 & 1 & -1 & 1 & -1 \\ 0 & 0 & 0 & 0 & -1 & 1 & -1 & 1 \end{bmatrix}\begin{bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}. \tag{5.112}$$

Adding the results for the two modes, we find the associated hourglass stiffness

$$\mathbf{k} = \frac{cb}{12a}\begin{bmatrix} 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0 \\ -1 & 1 & -1 & 1 & 0 & 0 & 0 & 0 \\ 1 & -1 & 1 & -1 & 0 & 0 & 0 & 0 \\ -1 & 1 & -1 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & -1 & 1 & -1 \\ 0 & 0 & 0 & 0 & -1 & 1 & -1 & 1 \\ 0 & 0 & 0 & 0 & 1 & -1 & 1 & -1 \\ 0 & 0 & 0 & 0 & -1 & 1 & -1 & 1 \end{bmatrix}. \tag{5.113}$$

A very small fraction of this stiffness is added to the singular element stiffness 5.103 to stabilize the hourglass modes without contributing significantly to the calculated displacements. ANSYS allows control of this factor by the parameter HGSTF. If the entire hourglass correction is added back, the stiffness matrix agrees with matrix 5.100 obtained by reduced integration on the λ terms. If the element is not rectangular, the definition of the hourglass modes must be altered.[6]

5.2.7 Example Using Improved Elements

The plane strain problem illustrated in Section 5.1.3 is analyzed using a 16 × 64 mesh using the Plane182 element of ANSYS. The result is shown in Table 5.4. This shows considerable improvement over the basic element as shown in Table 5.2 for ν near 0.5.

5.3 NUMERICAL INTEGRATION

Evaluation of the stiffness matrix involves integration over the region. This can be complicated for an arbitrary shape. It is therefore common to use numerical integration to obtain an approximate evaluation of the area and volume integrals.

TABLE 5.4
Comparison of Integration Methods

Element Option	ν	u_y	FEM/Exact
Full Integration ($\bar{\mathbf{B}}$)	0.3	0.951	0.997
Enhanced Strain	0.3	0.952	0.998
Reduced Integration	0.3	0.954	1.000
Full Integration ($\bar{\mathbf{B}}$)	0.499	0.802	0.998
Enhanced Strain	0.499	0.802	0.998
Reduced Integration	0.499	0.804	1.000

In order to evaluate the integral

$$\int_a^b f(x)\,\mathrm{d}x, \tag{5.114}$$

we choose several points x_i within the interval (a, b) and let $f(x_k)$ be denoted by f_k (Figure 5.13). We can approximate $f(x)$ in terms of the intermediate values f_k by some convenient curve fitting technique that makes integration easy. Then,

$$\int_a^b f(x)\,\mathrm{d}x = \sum_{n=1}^{N} W_n f_n. \tag{5.115}$$

The numerical coefficients W_n have been tabulated in reference works on numerical analysis for various choices of points x_k and interpolation functions. For multiple integrals, proceed by iterated integrals:

$$\begin{aligned}\int_c^d \int_a^b f(x,y)\,\mathrm{d}x\,\mathrm{d}y &= \int_c^d \sum_{n=1}^{N} f(X_n, y) W_n \mathrm{d}y \\ &= \sum_{n=1}^{N}\sum_{m=1}^{N} W_n W_m f(X_n, Y_m).\end{aligned} \tag{5.116}$$

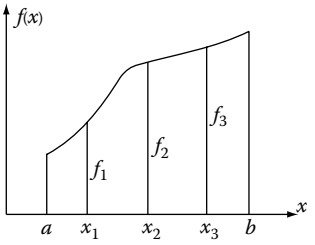


FIGURE 5.13 Numerical integration.

The most commonly used method of numerical integration for finite element applications is Gauss quadratures. In this method, the points x_n and coefficients W_n are chosen so that an exact result is obtained when $f(x)$ is a polynomial equal to 1, x, x^2, …, x^{2N-1}. This gives $2N$ conditions with which to determine the $2N$ quantities W_n and x_n.

For example, consider the case $N = 2$ and $-1 \le x \le 1$:

$$\int_{-1}^{1} \left(c_1 + c_2 x + c_3 x^2 + c_4 x^3\right) \mathrm{d}x \equiv W_1 f(x_1) + W_2 f(x_2). \tag{5.117}$$

From which,

$$2c_1 + \frac{2}{3} c_3 \equiv W_1 \left(c_1 + c_2 x_1 + c_3 x_1^2 + c_4 x_1^3\right) + W_2 \left(c_1 + c_2 x_2 + c_3 x_2^2 + c_4 x_2^3\right). \tag{5.118}$$

It follows that

$$W_1 = W_2 = 1, \quad x_1 = -\frac{1}{\sqrt{3}}, \quad x_2 = +\frac{1}{\sqrt{3}}. \tag{5.119}$$

For any function, we have the approximate formula:

$$\int_{-1}^{1} f(x)\,\mathrm{d}x = W_1 f(x_1) + W_2 f(x_2) = f(-0.577) + f(+0.577). \tag{5.120}$$

The factors have been tabulated[7] for a large number of choices of N. The results for one-, two-, three-, and four-point approximations for $a = -1$ and $b = +1$ are shown in Table 5.5. Note that the one-point integration is equivalent to taking the integrand as a constant equal to its value at the centroid.

TABLE 5.5
Weighting Factors

$N = 1$	$X_1 = 0$	$W_1 = 2$
$N = 2$	$X_1 = -0.57735027$	$W_1 = 1$
	$X_2 = +0.57735027$	$W_2 = 1$
$N = 3$	$X_1 = -0.77459$	$W_1 = 0.55555$
	$X_2 = 0$	$W_2 = 0.88888$
	$X_3 = +0.77459$	$W_3 = 0.55555$
$N = 4$	$X_1 = -0.86113$	$W_1 = 0.34785$
	$X_2 = -0.33998$	$W_2 = 0.65214$
	$X_3 = +0.33998$	$W_3 = 0.65214$
	$X_4 = +0.86113$	$W_4 = 0.34785$

5.4 COORDINATE TRANSFORMATIONS

To extend the previous results for rectangles, we need to introduce local coordinates for the element that are appropriate to the actual shape of the element. We gather here the formulas for transformation from the global rectangular Cartesian coordinates to local coordinates.

Consider a plane region covered by a rectangular Cartesian coordinate system $(x,y) = (x_1,x_2)$. Let us introduce a new (curvilinear) coordinate system by expressing x_α in terms of new parameters $(\xi,\eta) \equiv (\xi_1,\xi_2)$ by $x_\alpha = x_\alpha(\xi_1,\xi_2)$. We always choose a transformation that can be inverted: $\xi_\alpha = \xi_\alpha(x_1, x_2)$. Let

$$\frac{\partial x_\alpha}{\partial \xi_\beta} = J_{\alpha\beta}, \tag{5.121}$$

and keep in mind that $J_{\alpha\beta}$ is a function of (ξ,η). Furthermore, let

$$\frac{\partial \xi_\alpha}{\partial x_\beta} = J^*_{\alpha\beta}. \tag{5.122}$$

We then have (the summation convention applies to repeated indices):

$$\frac{\partial x_\alpha}{\partial \xi_\phi}\frac{\partial \xi_\phi}{\partial x_\beta} = \frac{\partial x_\alpha}{\partial x_\beta} = \delta_{\alpha\beta}. \tag{5.123}$$

Thus,

$$J_{\alpha\phi}\, J^*_{\phi\beta} = \delta_{\alpha\beta}. \tag{5.124}$$

In matrix notation

$$\mathbf{J}\mathbf{J}^* = \mathbf{1}. \tag{5.125}$$

The coordinate transformation is required to be invertible and therefore the determinant of J is nonzero:

$$J = \det \mathbf{J} \neq 0. \tag{5.126}$$

Integration over a region of the plane can be expressed in terms of the rectangular Cartesian system or the curvilinear coordinate system. For the rectangular Cartesian system, $\mathrm{d}a = \mathrm{d}x\mathrm{d}y$. For the curvilinear system, $da = J\mathrm{d}\xi\mathrm{d}\eta$. Therefore,

$$\iint_R f(x,y)\,\mathrm{d}x\,\mathrm{d}y = \iint_R f\,\mathrm{d}a = \iint_R fJ\,\mathrm{d}\xi\,\mathrm{d}\eta = \iint_R \bar{f}(\xi,\eta)\,\mathrm{d}\xi\,\mathrm{d}\eta. \tag{5.127}$$

If the coordinate transformation maps the region onto the square $-1 \le \xi \le 1$, $-1 \le \eta \le 1$, then integration formula 5.116 can be used:

$$\iint_R \bar{f}(\xi,\eta)\,d\xi\,d\eta = \sum_{n=1}^{N}\sum_{m=1}^{N} W_n W_m \bar{f}(\xi_n,\eta_m). \tag{5.128}$$

5.5 ISOPARAMETRIC QUADRILATERAL

Consider a general plane quadrilateral (Figure 5.14). Let us introduce a coordinate transformation such that the equation of the boundary segments has the simple form that ξ is constant or η is constant:

$$x_\alpha = \sum_{k=1}^{4} N_k(\xi,\eta) X_{\alpha k}, \tag{5.129}$$

where $(X_{1k},X_{2k}) \equiv (X_k,Y_k)$ are the (x,y) coordinates of node k. One such coordinate transformation is determined by the shape functions 5.10. We have, for example, when $\xi = +1$,

$$\begin{aligned} x &= \frac{1}{2}(1-\eta)X_2 + \frac{1}{2}(1+\eta)X_3, \\ y &= \frac{1}{2}(1-\eta)Y_2 + \frac{1}{2}(1+\eta)Y_3, \end{aligned} \tag{5.130}$$

which is the equation of the line 2–3. We must now find suitable shape functions as for the displacements within the quadrilateral. We will represent the components of displacement $(u,v) \equiv (u_1,v_2)$ in terms of their values $(U_{1k},U_{2k}) \equiv (U_k,V_k)$ at the four corners:

$$u_\alpha = \sum_{k=1}^{4} N_k(\xi,\eta) U_{\alpha k}. \tag{5.131}$$

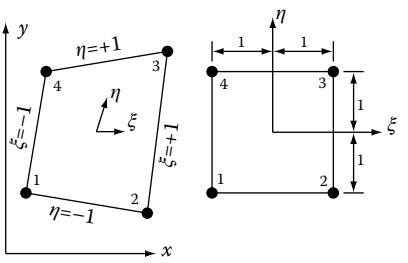


FIGURE 5.14 General quadrilateral.

Note that u_α and $U_{\alpha k}$ are components on the (x,y) system, not the (ξ,η) system. In matrix form,

$$\begin{matrix} U_{11} & U_{21} & U_{12} & U_{22} & U_{13} & U_{23} & U_{14} & U_{24} \\ U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 \\ D_1 & D_2 & D_3 & D_4 & D_5 & D_6 & D_7 & D_8 \end{matrix} \tag{5.132}$$

$$\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \mathbf{D}$$

or

$$\mathbf{u} = \mathbf{N}\mathbf{D}. \tag{5.133}$$

This curvilinear coordinate system is the natural choice that will simplify the task of finding suitable shape functions N_k. To achieve a continuous displacement field, it is only necessary to have u_α varying linearly with distance along an edge, and

$$N_k(\xi_m,\eta_m) = \delta_{km}. \tag{5.134}$$

Let us investigate the shape functions 5.10. Condition 5.134 is satisfied by this choice. It remains to be determined if the formula for u_α is linear along an edge. Along $\xi = 1$, for example,

$$\begin{aligned} u &= \frac{1}{2}(1-\eta)U_2 + \frac{1}{2}(1+\eta)U_3, \\ v &= \frac{1}{2}(1-\eta)V_2 + \frac{1}{2}(1+\eta)V_3. \end{aligned} \tag{5.135}$$

We see that u and v do vary linearly along the edge. Similar results hold for the other edges. Therefore, this choice of shape functions will generate a continuous displacement field for an assembly of quadrilaterals. When the same functions are used in the coordinate transformation 5.129 and the approximation for displacements 5.131, this procedure is known as the *isoparametric formulation*.[8]

We can now proceed to calculate the corresponding element stiffness matrix. From 5.131 and 5.122,

$$u_{\alpha,\beta} \equiv \frac{\partial u_\alpha}{\partial x_\beta} = \frac{\partial u_\alpha}{\partial \xi_\phi}\frac{\partial \xi_\phi}{\partial x_\beta} = \sum_k \sum_\phi N_{k\phi} J^*_{\phi\beta} U_{\alpha k}, \tag{5.136}$$

where the derivatives

$$N_{k\alpha} \equiv \frac{\partial N_k}{\partial \xi_\alpha} \tag{5.137}$$

are determined from 5.12. From 2.27,

$$\begin{aligned} 2\varepsilon_{\alpha\beta} &= \sum_{k,\phi} \left(N_{k\phi} J^*_{\phi\beta} U_{\alpha k} + N_{k\phi} J^*_{\phi\alpha} U_{\beta k} \right) \\ &= \sum_{k,\phi,\theta} \left(N_{k\phi} J^*_{\phi\beta} \delta_{\theta\alpha} U_{\theta k} + N_{k\phi} J^*_{\phi\alpha} \delta_{\theta\beta} U_{\theta k} \right) \\ &= \sum_{k,\phi,\theta} \left(N_{k\phi} J^*_{\phi\beta} \delta_{\theta\alpha} + N_{k\phi} J^*_{\phi\alpha} \delta_{\theta\beta} \right) U_{\theta k} \\ &= \sum_{\theta.k} A_{\alpha\beta\theta k} U_{\theta k}, \end{aligned} \tag{5.138}$$

where

$$A_{\alpha\beta\theta k} = \sum_{k,\phi} (N_{k\phi} J^*_{\phi\beta} \delta_{\theta\alpha} + N_{k\phi} J^*_{\phi\alpha} \delta_{\theta\beta}). \tag{5.139}$$

In matrix form,

$$\begin{matrix} & U_{11} & U_{21} & U_{12} & U_{22} & U_{13} & U_{23} & U_{14} & U_{24} \\ & U_1 & V_1 & U_2 & V_2 & U_3 & V_3 & U_4 & V_4 \\ & D_1 & D_2 & D_3 & D_4 & D_5 & D_6 & D_7 & D_8 \end{matrix}$$

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{bmatrix} = \begin{bmatrix} A_{1111} & A_{1121} & A_{1112} & A_{1122} & A_{1113} & A_{1123} & A_{1114} & A_{1124} \\ A_{2211} & A_{2221} & A_{2212} & A_{2222} & A_{2213} & A_{12223} & A_{2214} & A_{2224} \\ A_{1211} + A_{2111} & & & \text{etc.} & & & & \end{bmatrix} [D_n]. \tag{5.140}$$

In matrix notation,

$$\boldsymbol{\varepsilon} = \mathbf{AD}. \tag{5.141}$$

By 3.23 and 3.24, the stiffness matrix for the element is

$$\mathbf{k} = \int_R \mathbf{A}^{\mathrm{T}} \mathbf{C} \mathbf{A} \, \mathrm{d}a. \tag{5.142}$$

We need the Jacobian $J_{\alpha\beta} = \partial x_\alpha / \partial \xi_\beta$:

$$
\begin{aligned}
J_{11} &= \frac{\partial x_1}{\partial \xi_1} = \frac{\partial x}{\partial \xi} = \sum_k \frac{\partial N_k}{\partial \xi} X_k = \frac{1}{4} \sum_k X_k \xi_k (1 + \eta_k \eta), \\
J_{22} &= \frac{\partial x_2}{\partial \xi_2} = \frac{\partial y}{\partial \eta} = \sum_k \frac{\partial N_k}{\partial \eta} Y_k = \frac{1}{4} \sum_k Y_k \eta_k (1 + \xi_k \xi), \\
J_{12} &= \frac{1}{4} \sum_k X_k \eta_k (1 + \xi_k \xi), \\
J_{21} &= \frac{1}{4} \sum_k Y_k \xi_k (1 + \eta_k \eta),
\end{aligned}
\tag{5.143}
$$

where ξ_k and η_k are nodal coordinates. Thus, $J_{\alpha\beta}$ is a polynomial, but $J^*_{\alpha\beta}$ is not, and the integration is complicated. Numerical integration is therefore used. The integral over the general quadrilateral is accomplished by using the coordinate transformation to (ξ,η) as in 5.127 and using numerical integration as in 5.128.

$$
\mathbf{k} = \iint \mathbf{f}(\xi, \eta)\, \mathrm{d}\xi\, \mathrm{d}\eta = \sum_i \sum_j \mathbf{f}(\xi_i, \eta_j) W_i W_j, \tag{5.144}
$$

where ξ_k and η_k are coordinates of the IPs.

The foregoing relations are expressed in the usual matrix form with the following correspondence (the numbering is chosen to match the FORTRAN storage convention):

$$
\begin{bmatrix} D_1 \\ D_2 \\ D_3 \\ \vdots \end{bmatrix} = \begin{bmatrix} U_1 \\ V_1 \\ U_2 \\ \vdots \end{bmatrix} = \begin{bmatrix} U_{11} \\ U_{21} \\ U_{12} \\ \vdots \end{bmatrix}. \tag{5.145}
$$

The procedures used in this section to find a stiffness matrix for the quadrilateral as soon as one is known for the rectangle, provide a general method that can be used for other types of elements:

(1) Find the shape functions for the rectangle.
(2) Introduce natural coordinates (ξ, η) for the quadrilateral as the inverse of the coordinate transformation 5.129.
(3) Shape functions for the displacements within the quadrilateral are given by 5.131.

(4) Use the general relations to obtain the stiffness matrix for the quadrilateral that involves the integral over the quadrilateral region.
(5) Use numerical integration to evaluate the integral.

5.5.1 Wilson–Taylor Element

For this element, 5.131 is replaced by

$$u_\alpha = \sum_{k=1}^{4} N_k(\xi,\eta)U_{\alpha k} + \sum_{a=1}^{4} N_{\alpha a}^{\text{ext}} d_a \tag{5.146}$$

where

$$\begin{aligned} N_{11}^{\text{ext}} &= N_{21}^{\text{ext}} = 1-\xi^2, \\ N_{12}^{\text{ext}} &= N_{22}^{\text{ext}} = 1-\eta^2. \end{aligned} \tag{5.147}$$

Equation 5.138 is replaced by

$$2\varepsilon_{\alpha\beta} = \sum_{\theta.k} A_{\alpha\beta\theta k} U_{\theta k} + \sum_{a=1}^{4} B_{\alpha\beta a} d_a, \tag{5.148}$$

where

$$B_{\alpha\beta a} = \frac{\partial N_{\alpha a}^{\text{ext}}}{\partial x_\beta} + \frac{\partial N_{\beta a}^{\text{ext}}}{\partial x_\alpha} = \frac{\partial N_{\alpha a}^{\text{ext}}}{\partial \xi_\theta} J_{\theta\beta}^{*} + \frac{\partial N_{\beta a}^{\text{ext}}}{\partial \xi_\theta} J_{\theta\alpha}^{*}. \tag{5.149}$$

This expression is simplified by evaluating the Jacobean terms $J_{\alpha\beta}^{*}$ at the centroid. It is shown below that the patch test will then be satisfied by the element.

5.5.2 Three-Node Triangle as a Special Case of Rectangle

The constant strain triangle can be obtained from the four-node quadrilateral as the special case when two nodes coincide (Figure 5.15). Collapsing node 4 onto node 1, the coordinate transformation 5.129 with $X_1 = X_4$ and $Y_1 = Y_4$ becomes

$$\begin{aligned} x &= \frac{1}{2}(1-\xi)X_1 + \frac{1}{4}(1+\xi)(1-\eta)X_2 + \frac{1}{4}(1+\xi)(1+\eta)X_3, \\ y &= \frac{1}{2}(1-\xi)Y_1 + \frac{1}{4}(1+\xi)(1-\eta)Y_2 + \frac{1}{4}(1+\xi)(1+\eta)Y_3. \end{aligned} \tag{5.150}$$

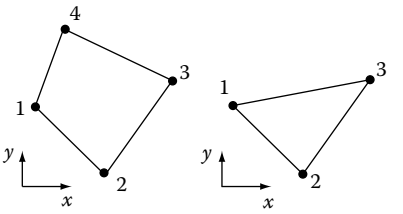


FIGURE 5.15 Collapse into a triangle.

This induces shape functions for the triangle:

$$N_1 = \frac{1}{2}(1-\xi), \quad N_2 = \frac{1}{4}(1+\xi)(1-\eta), \quad N_3 = \frac{1}{4}(1+\xi)(1+\eta). \tag{5.151}$$

Note that

$$N_1 + N_2 + N_3 = 1. \tag{5.152}$$

That is, the shape functions satisfy the relation

$$\begin{bmatrix} 1 \\ x \\ y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ X_1 & X_2 & X_3 \\ Y_1 & Y_2 & Y_3 \end{bmatrix} \begin{bmatrix} N_1 \\ N_2 \\ N_3 \end{bmatrix}. \tag{5.153}$$

Therefore, by 4.4,

$$\begin{bmatrix} N_1 \\ N_2 \\ N_3 \end{bmatrix} = (\mathbf{\Phi}^{-1})^{\mathrm{T}} \begin{bmatrix} 1 \\ x \\ y \end{bmatrix} = \begin{bmatrix} \zeta_1 \\ \zeta_2 \\ \zeta_3 \end{bmatrix} \tag{5.154}$$

where ζ_i represents the triangular coordinates. These are the shape functions for the constant strain triangle.

5.6 EIGHT-NODE QUADRILATERAL

The shape functions for the four-node rectangle described in Section 5.1 can be extended to an eight-note rectangle in which the additional nodes are placed at the midpoints of the sides, as in Figure 5.16. The natural coordinates (ξ,η) as defined in Section 5.1 will be used here. The formulas for the x and y components of displacements in the element have the form

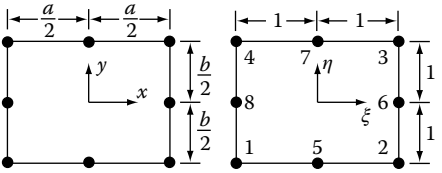


FIGURE 5.16 Eight-node rectangle.

$$u = \sum_{i=1}^{8} N_i(\xi,\eta)U_i,$$
$$v = \sum_{i=1}^{8} N_i(\xi,\eta)V_i. \tag{5.155}$$

The shape functions for the corners must be modified so that they are zero at the side nodes, and additional shape functions must be found for the side nodes. Suitable shape functions that can be found by trial and error are as follows.

$$N_i = \frac{1}{4}(1+\xi_i\xi)(1+\eta_i\eta)(\xi_i\xi+\eta_i\eta-1),\ \ i=1,2,3,4.$$
$$N_i = \frac{1}{2}\left(1-\xi^2\right)(1+\eta_i\eta),\ \ i=5,7. \tag{5.156}$$
$$N_i = \frac{1}{2}(1+\xi_i\xi)\left(1-\eta^2\right),\ \ i=6,8.$$

One can see by inspection that the shape functions have the property,

$$N_i(\xi_j,\eta_j) = \delta_{ij}, \tag{5.157}$$

as required. The continuity between elements now has to be investigated. On the side $\xi = 1$, for example,

$$u = \frac{1}{2}\left(-\eta+\eta^2\right)U_2 + \frac{1}{2}\left(\eta+\eta^2\right)U_3 + \left(1-\eta^2\right)U_6. \tag{5.158}$$

That is, u is a quadratic function. Since a quadratic function is uniquely determined by its value at three points, two elements sharing the three nodes on that side will have matching displacements along that side. Similar calculations along the other sides show that continuity between elements is achieved for these shape functions. If one expands the representation for u, the result is of the form

$$u = c_1 + c_2\xi + c_3\eta + c_4\xi^2 + c_5\xi\eta + c_6\eta^2 + c_7\xi^2\eta + c_8\eta^2\xi, \tag{5.159}$$

where coefficients c_i are combinations of the nodal values. That is, the representation of u is an incomplete cubic polynomial that utilizes the bold-face terms in Table 5.6.

TABLE 5.6
Polynomial Terms

1	$\boldsymbol{\xi}$	$\boldsymbol{\xi^2}$	ξ^3
η	$\boldsymbol{\xi\eta}$	$\boldsymbol{\xi^2\eta}$	$\xi^3\eta$
$\boldsymbol{\eta^2}$	$\boldsymbol{\xi\eta^2}$	$\xi^2\eta^2$	$\xi^3\eta$
η^3	$\xi\eta^3$	$\xi^2\eta^3$	$\xi^3\eta^3$

The important thing is that all of the linear terms are present so that constant strain can be represented exactly. In fact, the displacements are complete in the second-order terms, so that the representation for the strains is complete in the linear terms. The strains for an element are given by

$$\varepsilon_{11} = \frac{2}{a}\sum_{k=1}^{8}\frac{\partial N_k}{\partial \xi}U_k = \frac{2}{a}\left\lfloor \frac{\partial N_k}{\partial \xi}\right\rfloor\left\{U_k\right\}. \tag{5.160}$$

$$\varepsilon_{22} = \frac{2}{b}\sum_{k=1}^{8}\frac{\partial N_k}{\partial \eta}V_k = \frac{2}{b}\left\lfloor \frac{\partial N_k}{\partial \eta}\right\rfloor\left\{V_k\right\}. \tag{5.161}$$

$$\gamma_{12} = \frac{2}{b}\left\lfloor \frac{\partial N_k}{\partial \eta}\right\rfloor\left\{U_k\right\} + \frac{2}{a}\left\lfloor \frac{\partial N_k}{\partial \xi}\right\rfloor\left\{V_k\right\}. \tag{5.162}$$

Thus,

$$\mathbf{A} = \frac{2}{ab}\begin{bmatrix} & \mathbf{U} & \mathbf{V} \\ & b\left\lfloor \dfrac{\partial N_k}{\partial \xi}\right\rfloor & \mathbf{0} \\ & \mathbf{0} & a\left\lfloor \dfrac{\partial N_k}{\partial \eta}\right\rfloor \\ & a\left\lfloor \dfrac{\partial N_k}{\partial \eta}\right\rfloor & b\left\lfloor \dfrac{\partial N_k}{\partial \xi}\right\rfloor \end{bmatrix}. \tag{5.163}$$

Using 2×2 numerical integration, the stiffness matrix of the element is

$$\mathbf{k} = \iint \mathbf{A}^{\mathrm{T}}\mathbf{C}\mathbf{A}\,\mathrm{d}a = \frac{ab}{4}\int_{-1}^{1}\int_{-1}^{1}\mathbf{f}(\xi,\eta)\,\mathrm{d}\xi\,\mathrm{d}\eta = \frac{ab}{4}\sum_{i=1}^{4}\mathbf{f}(\xi_i,\eta_i) \tag{5.164}$$

where $\mathbf{f} = \mathbf{A}^{\mathrm{T}}\mathbf{C}\mathbf{A}$ and $(\xi_i,\eta_i) = (\pm 1/\sqrt{3}, \pm 1/\sqrt{3})$ are the coordinates of the IPs. The resulting matrix for $a = b$ and $\nu = 1/3$ is

$$\mathbf{K} = \begin{array}{c} \begin{matrix} \quad\mathbf{U} & \quad\mathbf{V} \end{matrix} \\ \begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{uv} \\ \mathbf{K}_{uv}^{\mathrm{T}} & \mathbf{K}_{vv} \end{bmatrix} \end{array} \tag{5.165}$$

where

$$\mathbf{K}_{uu} = \frac{20\mu}{9}\begin{bmatrix} 1.000 & 0.525 & 0.500 & 0.375 & -1.150 & -0.350 & -0.650 & -0.250 \\ & 1.000 & 0.375 & 0.500 & -1.150 & -0.250 & -0.650 & -0.350 \\ & & 1.000 & 0.525 & -0.650 & -0.250 & -1.150 & -0.350 \\ & & & 1.000 & -0.650 & -0.350 & -1.150 & -0.250 \\ & & & & 2.600 & 0 & 1.000 & 0 \\ & \text{Sym} & & & & 1.400 & 0 & -0.200 \\ & & & & & 0 & 2.600 & 0 \\ & & & & & & & 1.400 \end{bmatrix}, \tag{5.166}$$

$$\mathbf{K}_{vv} = \frac{20\mu}{9}\begin{bmatrix} 1.000 & 0.375 & 0.500 & 0.525 & -0.250 & -0.650 & -0.350 & -1.150 \\ & 1.000 & 0.525 & 0.500 & -0.250 & -1.150 & -0.350 & -0.650 \\ & & 1.000 & 0.375 & -0.350 & -1.150 & -0.250 & -0.650 \\ & & & 1.000 & -0.350 & -0.650 & -0.250 & -1.150 \\ & & & & 1.400 & 0 & -0.200 & 0 \\ & \text{Sym} & & & & 2.600 & 0 & 1.000 \\ & & & & & & 1.400 & 0 \\ & & & & & & & 2.600 \end{bmatrix}, \tag{5.167}$$

$$\mathbf{K}_{uv} = \frac{20\mu}{9}\begin{bmatrix} 0.425 & 0 & 0.175 & 0 & -0.200 & -0.100 & -0.100 & -0.200 \\ 0 & -0.425 & 0 & -0.175 & 0.200 & 0.200 & 0.100 & 0.100 \\ 0.175 & 0 & 0.425 & 0 & -0.100 & -0.200 & -0.200 & -0.100 \\ 0 & -0.175 & 0 & -0.425 & 0.100 & 0.100 & 0.200 & 0.200 \\ -0.200 & 0.200 & -0.100 & 0.100 & 0 & -0.400 & 0 & 0.400 \\ -0.100 & 0.200 & -0.200 & 0.100 & -0.400 & 0 & 0.400 & 0 \\ -0.100 & 0.100 & -0.200 & 0.200 & 0 & 0.400 & 0 & -0.400 \\ -0.200 & 0.100 & -0.100 & 0.200 & 0.400 & 0 & -0.400 & 0 \end{bmatrix}. \tag{5.168}$$

This agrees with the matrix for ANSYS element Plane183.

In order to have a further check on the implementation code, let us evaluate the stress matrices for a single element. Let $a = 1$, $b = 1$, $\mu = 1$, $\nu = 1/3$, $E = 8/3$. For plane stress,

$$\begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{12} \end{bmatrix} = \begin{bmatrix} 3 & 1 & 0 \\ 1 & 3 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{bmatrix}. \tag{5.169}$$

The strains are expressed by 5.160–5.162. At the centroid, $\xi = 0$, $\eta = 0$,

$$\begin{aligned} \varepsilon_{11} &= \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 \end{bmatrix} \{U_k\}, \\ \varepsilon_{22} &= \begin{bmatrix} 0 & 0 & 0 & 0 & -1 & 0 & 1 & 0 \end{bmatrix} \{V_k\}, \\ \gamma_{12} &= \begin{bmatrix} 0 & 0 & 0 & 0 & -1 & 0 & 1 & 0 \end{bmatrix} \{U_k\} \\ &\quad + \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 1 & 0 & -1 \end{bmatrix} \{V_k\}. \end{aligned} \tag{5.170}$$

The stress at the centroid is therefore given by

$$\begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{12} \end{bmatrix} = \begin{bmatrix} 0 & 3 & 0 & -3 & -1 & 0 & 1 & 0 \\ 0 & 1 & 0 & -1 & -3 & 0 & 3 & 0 \\ 1 & 0 & 1 & 0 & 0 & 1 & 0 & 1 \end{bmatrix} \begin{bmatrix} U_5 \\ U_6 \\ U_7 \\ U_8 \\ V_5 \\ V_6 \\ V_7 \\ V_8 \end{bmatrix}. \tag{5.171}$$

At node 3, $\xi = 1$, $\eta = 1$,

$$\begin{aligned} \varepsilon_{11} &= \begin{bmatrix} 0 & 0 & 3 & 1 & 0 & -4 & 0 & 0 \end{bmatrix} \{U_k\}, \\ \varepsilon_{22} &= \begin{bmatrix} 0 & 1 & 3 & 0 & 0 & 0 & -4 & 0 \end{bmatrix} \{V_k\}, \\ \gamma_{12} &= \begin{bmatrix} 0 & 1 & 3 & 0 & 0 & 0 & -4 & 0 \end{bmatrix} \{U_k\} \\ &\quad + \begin{bmatrix} 0 & 0 & 3 & 1 & 0 & -4 & 0 & 0 \end{bmatrix} \{V_k\}. \end{aligned} \tag{5.172}$$

At the IPs, $\xi = \pm 1/\sqrt{3}$, $\eta = \pm 1/\sqrt{3}$, for $U_1 = 1$ and the remaining DOF zero, the stresses are listed in Table 5.7.

TABLE 5.7
Integration Point Stress

ξ	η	τ_{11}	τ_{22}	τ_{12}
0.577350269	0.577350269	1.098076211	0.366025404	0.366025404
0.577350269	–0.577350269	1.366025404	0.455341801	–0.122008468
–0.577350269	0.577350269	–0.366025404	–0.122008468	0.455341801
–0.577350269	–0.577350269	–4.098076211	–1.366025404	–1.366025404

TABLE 5.8
Extrapolated Nodal Stress for $U_1 = 1$

Node	τ_{11}	τ_{22}	τ_{12}
1	–8	–2.6667	–2.6667
2	4	4/3	1/3
3	1	1/3	1/3
4	1	1/3	4/3

ANSYS does not use formula 5.172 to calculate the nodal stress. Instead, the shape functions 5.10 for the four-node quadrilateral are used for each component σ of stress,

$$\sigma(x,y) = \sum_{k=1}^{4} \frac{1}{4}\left(1+3\xi_k\xi\right)\left(1+3\eta_k\eta\right)\sigma_k, \tag{5.173}$$

where ξ_K and η_K are the coordinates of the IP, and σ_K is the value of σ at the IP. The extrapolated IP stresses are shown in Table 5.8. With ERESX = Yes, the default value, the element stresses are extrapolated to the nodes by 5.173. With ERESX = No, the IP stresses are copied to the nodal points (ANSYS: Solution > Load Step Options > Output Ctrls > Integration Pt).

5.6.1 Nodal Loads

The energy equivalent node forces for distributed loads are not so easy to anticipate. They are not what one would expect from statics. One must evaluate the integrals in 3.12 using the shape functions 5.156. In the case of a load in the x direction that is distributed uniformly over the side 2–6–3, one finds that 1/6 of the total load acts at the corner nodes and 2/3 of the load acts at the mid-side node (Figure 5.17). The nodal forces for a linearly varying load in the x direction are shown in Figure 5.18.

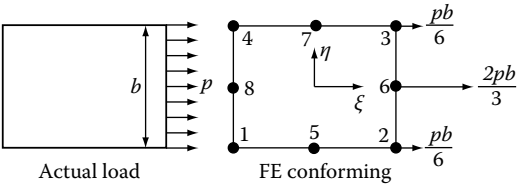


FIGURE 5.17 Node forces for a uniform load.

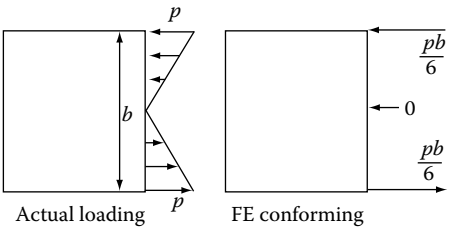


FIGURE 5.18 Nodal forces for linear load.

5.6.2 Plane Stress Example: Pure Bending

In order to test a finite element implementation, we need a nontrivial problem for which the exact solution is known. The problem diagrammed in Figure 5.19 has the following solution and can be used as a test problem.

$$
\begin{aligned}
u(x,y) &= -\frac{pxy}{E}, \\
v(x,y) &= \frac{p}{2E}\left(x^2 + \nu y^2\right), \\
\tau_{11} &= -py, \\
\tau_{22} &= 0, \\
\tau_{12} &= 0.
\end{aligned}
\tag{5.174}
$$

Since the element displacement field (5.6.5) is complete in second-order terms, a one-element model will provide an exact solution if exact integration is used. Two-point integration also gives an exact result.

5.6.3 Plane Stress Example: Bending with Shear

A second example (Figure 5.20) for which the exact solution is known was analyzed in Section 5.1.3 using the four-node element. The exact solution 2.41 for plane stress or plane strain is

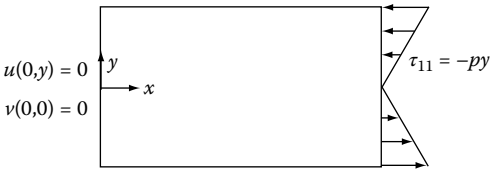


FIGURE 5.19 Plane stress bending.

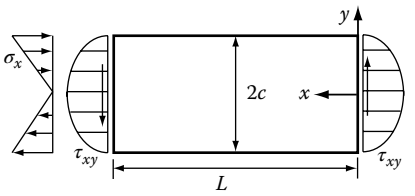


FIGURE 5.20 Plane stress bending with shear.

$$\sigma_x = \frac{3Pxy}{2c^3},$$
$$\sigma_y = 0, \tag{5.175}$$
$$\tau_{xy} = \frac{3P}{4c}\left(1 - \frac{y^2}{c^2}\right).$$

The displacements for plane stress are given in 2.42.

We will now solve this problem using the eight-node element. The finite element solution is not exact in this case because of the cubic variation in displacements as shown in 2.142. Since there is antisymmetry about the x axis, we only need to consider the upper half of the body and the antisymmetry conditions on the x axis as shown in Figure 5.21. Note the change in coordinates between Figures 5.20 and 5.21. For $c = 1$, $L = 10$, $P = 20$, the loads are as shown. The antisymmetry conditions require that the x displacement and the normal stress vector are zero along the x axis. To prevent rigid body displacement, we can set the y displacement to zero at the origin and set the x displacement to zero at the upper left corner. With these support conditions, the displacement at the centerline of the free end ($x = L$, $y = 0$) is given by 2.44:

$$v(L,0) = \frac{PL^3}{3EI}\left(1 + \frac{(4+5\nu)}{2}\frac{c^2}{L^2}\right) = 1.0283 \tag{5.176}$$

for $E = 10{,}000$, $\nu = 1/3$. Using a 50 (wide) by 10 (high) mesh of 500 eight-node elements, we obtain a displacement of 1.0243, an error of 0.39%. The calculated nodal stress is shown in Figures 5.22 and 5.23. The maximum $\sigma_x = 291$, an error of 3%. The maximum $\tau_{xy} = 15.01$, an error of 0.07%. The distribution closely follows the exact solution 5.175 except near the corner where the particular support condition used perturbs the numerical solution.

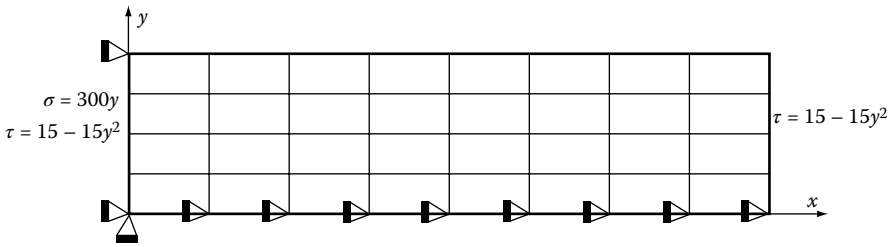


FIGURE 5.21 Bending with shear: finite element model.

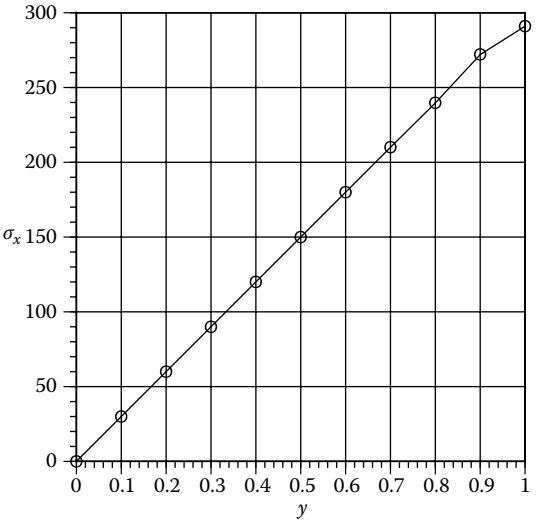


FIGURE 5.22 Bending with shear: σ_x at supported end.

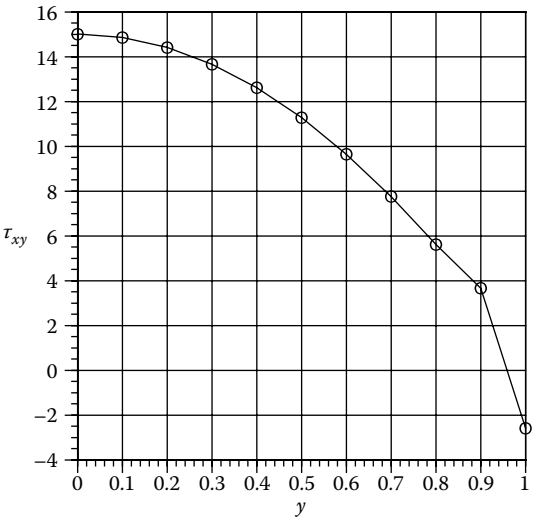


FIGURE 5.23 Bending with shear: τ_{xy} at supported end.

The support $u(0,c) = 0$, which is necessary to prevent rigid rotation of the body, has a local effect on the finite element analysis even though it has no effect on the exact solution.

5.6.4 Plane Stress Example: Short Beam

Let us consider the example problem shown in Figure 5.24. The representation for the four-node rectangle of Section 5.1 is an incomplete quadratic form, and the representation for the strains is therefore an incomplete linear form. One would expect that significantly better answers would be obtained for the same number of rectangles if one uses the eight-node element that is complete in the linear strain. However, there are then more nodes and more computational effort is required to achieve this improved result. For this example problem, the corner displacement obtained with the eight-node element is shown in Table 5.9.

These results should be compared to those in Table 5.3. The accuracy for a corresponding number of nodes is compared in Figure 5.25.

5.6.5 General Quadrilateral Element

The shape functions 5.156 can be used to generate an eight-node "quadrilateral" with curved sides via the isoparametric analysis developed in Section 5.5. The basic

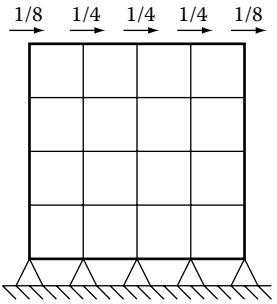


FIGURE 5.24 Plane stress problem: short beam.

TABLE 5.9
Results for an Eight-Node Rectangle

Segments per Side	No. of Nodes	No. of Rectangles	x Displacement of the Corner
1	8	1	6.4508
2	21	4	7.0731
4	65	16	7.3123
5	96	25	7.3447
10	341	100	7.3990
20	1281	400	7.4228

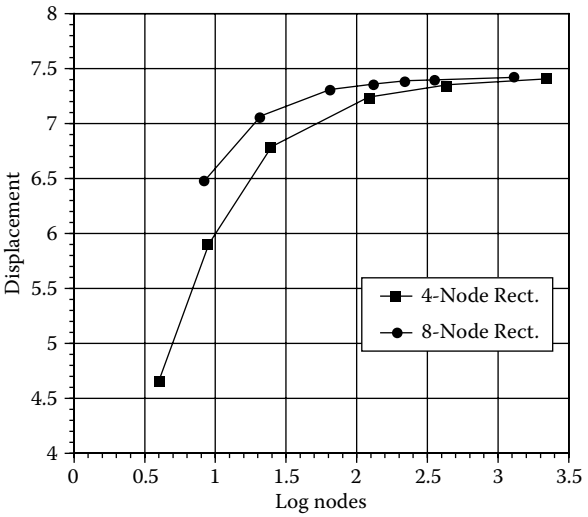


FIGURE 5.25 Convergence for short beam.

equations have the same form as in Chapter 15, but with the range of summation for the node index being 8 in place of 4.

The natural coordinate system (ξ,η) defined by the inverse of relation 5.155 is truly a curved system and numerical integration is a necessity. The order of integration must be high enough to provide accuracy without being too costly in computing time. Integration of order two in ξ and η, four IPs, is most commonly used: $\xi = \pm 1/\sqrt{3}, \eta = \pm 1/\sqrt{3}$.

The sides of the general element are curved. Consider, for example, the edge 2–6–3, where $\xi = 1$. The mapping defined by the shape functions becomes

$$\begin{aligned} x &= -\frac{1}{2}(1-\eta)\eta X_2 + \left(1-\eta^2\right)X_6 + \frac{1}{2}(1+\eta)\eta X_3, \\ y &= -\frac{1}{2}(1-\eta)\eta Y_2 + \left(1-\eta^2\right)Y_6 + \frac{1}{2}(1+\eta)\eta Y_3. \end{aligned} \tag{5.177}$$

Since the coordinates of points 2, 3, and 6 can be chosen independently, we see that the side 2–6–3 is in general a quadratic curve determined by the coordinates of the three nodes. This offers some advantage in fitting curved boundaries with a small number of elements, but there is a cost in increased complexity of the model.

5.7 EIGHT-NODE BLOCK

The 3D extension of the four-node rectangle is an eight-node brick element (Figure 5.26). The side lengths and local rectangular Cartesian coordinates are

$$-\frac{a}{2} \le x \le \frac{a}{2}, \quad -\frac{b}{2} \le y \le \frac{b}{2}, \quad -\frac{c}{2} \le z \le \frac{c}{2}. \tag{5.178}$$

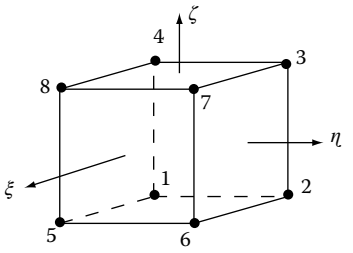


FIGURE 5.26 Eight-node solid.

The serendipity shape functions are

$$N_i = \frac{1}{8}(1+\xi_i\xi)(1+\eta_i\eta)(1+\zeta_i\zeta). \tag{5.179}$$

Since there are 3 DOF per node, there are 24 DOF per element. The stiffness matrix is 24 × 24. Let

$$\mathbf{A} = [A_k] = [\partial N_k/\partial\xi], \quad \mathbf{B} = [B_k] = [\partial N_k/\partial\eta], \quad \mathbf{C} = [C_k] = [\partial N_k/\partial\zeta]. \tag{5.180}$$

The components of the strain tensor 2.1 are

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{12} \\ 2\varepsilon_{23} \\ 2\varepsilon_{31} \end{bmatrix} = \begin{bmatrix} \frac{2}{a}\mathbf{A} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \frac{2}{b}\mathbf{B} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \frac{2}{c}\mathbf{C} \\ \frac{2}{b}\mathbf{B} & \frac{2}{a}\mathbf{A} & \mathbf{0} \\ \mathbf{0} & \frac{2}{c}\mathbf{C} & \frac{2}{b}\mathbf{B} \\ \frac{2}{c}\mathbf{C} & \mathbf{0} & \frac{2}{a}\mathbf{A} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \\ \mathbf{W} \end{bmatrix}. \tag{5.181}$$

The elastic matrix 2.82 is

$$\mathbf{C} = \mu \begin{bmatrix} \alpha & \beta & \beta & 0 & 0 & 0 \\ \beta & \alpha & \beta & 0 & 0 & 0 \\ \beta & \beta & \alpha & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}, \quad \alpha = \frac{2(1-\nu)}{1-2\nu}, \quad \beta = \frac{2\nu}{1-2\nu}. \tag{5.182}$$

The stiffness matrix 3.24 is therefore

$$\mathbf{k} = \frac{\mu abc}{2}\begin{bmatrix} \mathbf{U} & \mathbf{V} & \mathbf{W} \\ \frac{\alpha}{a^2}\mathbf{A}_{11} + \frac{1}{b^2}\mathbf{A}_{22} + \frac{1}{c^2}\mathbf{A}_{33} & & \text{symmetric} \\ \frac{\beta}{ab}\mathbf{A}_{21} & \frac{\alpha}{b^2}\mathbf{A}_{22} + \frac{1}{c^2}\mathbf{A}_{33} + \frac{1}{a^2}\mathbf{A}_{11} & \\ \frac{\beta}{ab}\mathbf{A}_{31} & \frac{\beta}{ab}\mathbf{A}_{32} & \frac{\alpha}{c^2}\mathbf{A}_{33} + \frac{1}{a^2}\mathbf{A}_{11} + \frac{1}{b^2}\mathbf{A}_{22} \end{bmatrix}, \tag{5.183}$$

where

$$\begin{aligned}
\mathbf{A}_{11} &= \iiint \mathbf{A}^{\mathrm{T}}\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta\,\mathrm{d}\zeta = \frac{1}{72}\left[\xi_k\xi_m(3+\eta_k\eta_m)(3+\zeta_k\zeta_m)\right] \\
\mathbf{A}_{22} &= \iiint \mathbf{B}^{\mathrm{T}}\mathbf{B}\,\mathrm{d}\xi\,\mathrm{d}\eta\,\mathrm{d}\zeta = \frac{1}{72}\left[\eta_k\eta_m(3+\zeta_k\zeta_m)(3+\xi_k\xi_m)\right] \\
\mathbf{A}_{33} &= \iiint \mathbf{C}^{\mathrm{T}}\mathbf{C}\,\mathrm{d}\xi\,\mathrm{d}\eta\,\mathrm{d}\zeta = \frac{1}{72}\left[\zeta_k\zeta_m(3+\xi_k\xi_m)(3+\eta_k\eta_m)\right] \\
\mathbf{A}_{12} &= \mathbf{A}_{21}^{T} = \iiint \mathbf{A}^{\mathrm{T}}\mathbf{B}\,\mathrm{d}\xi\,\mathrm{d}\eta\,\mathrm{d}\zeta = \frac{1}{24}\left[\xi_k\eta_m(3+\zeta_k\zeta_m)\right] \\
\mathbf{A}_{23} &= \mathbf{A}_{32}^{T} = \iiint \mathbf{B}^{\mathrm{T}}\mathbf{C}\,\mathrm{d}\xi\,\mathrm{d}\eta\,\mathrm{d}\zeta = \frac{1}{24}\left[\eta_k\zeta_m(3+\xi_k\xi_m)\right] \\
\mathbf{A}_{31} &= \mathbf{A}_{13}^{T} = \iiint \mathbf{C}^{\mathrm{T}}\mathbf{A}\,\mathrm{d}\xi\,\mathrm{d}\eta\,\mathrm{d}\zeta = \frac{1}{24}\left[\zeta_k\xi_m(3+\eta_k\eta_m)\right].
\end{aligned} \tag{5.184}$$

For the case $a = b = c = 2$, $\nu = 1/3$, $\alpha = 4$, $\beta = 2$, the first column of the element stiffness matrix is

$$[FX_k] = \frac{\mu}{24}\begin{bmatrix} 32 \\ -16 \\ -12 \\ 8 \\ 8 \\ -12 \\ -8 \\ 0 \end{bmatrix}, \quad [FY_k] = \frac{\mu}{24}\begin{bmatrix} 12 \\ 4 \\ -12 \\ -4 \\ 6 \\ 2 \\ -6 \\ -2 \end{bmatrix}, \quad [FZ_k] = \frac{\mu}{24}\begin{bmatrix} 12 \\ 4 \\ 2 \\ 6 \\ -4 \\ -12 \\ -6 \\ -2 \end{bmatrix}. \tag{5.185}$$

This agrees with ANSYS element SOLID45 without extra DOFs. Although this element is very instructive and valuable for historical reasons, ANSYS is phasing it out. I will continue to refer to it as a synonym for an analysis based on the developments of this section.

5.8 TWENTY-NODE SOLID

The eight-node quadrilateral becomes a 20-node solid (Figure 5.27). The shape functions are as follows.

$$\begin{aligned}
&\text{Nodes } 1-8\text{: } N_i = \frac{1}{8}(1+\xi_i\xi)(1+\eta_i\eta)(1+\zeta_i\zeta)(\xi_i\xi+\eta_i\eta+\zeta_i\zeta-2).\\
&\text{Nodes } 9,11,13,15\text{: } N_i = \frac{1}{4}(1+\xi_i\xi)(1-\eta^2)(1+\zeta_i\zeta).\\
&\text{Nodes } 10,12,14,16\text{: } N_i = \frac{1}{4}(1+\xi_i\xi)(1+\eta_i\eta)(1-\zeta^2).\\
&\text{Nodes } 17,18,19,20\text{: } N_i = \frac{1}{4}(1-\xi^2)(1+\eta_i\eta)(1+\zeta_i\zeta).
\end{aligned} \tag{5.186}$$

This is the basis for the ANSYS element 186.

Many other elements have been developed for both two and three dimensions.[9] Those included here are the most commonly used.

5.9 SINGULARITY ELEMENT

We have seen that singularities in the solution to boundary value problems may occur when there are discontinuities in the boundary conditions. The solution for stress near the singularity is not very good if we use standard elements. The solution can be improved by using elements with the theoretical singularity built into them. As an example, let us consider a six-node rectangle for which two of the side nodes

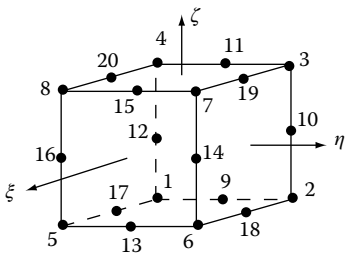


FIGURE 5.27 Twenty-node solid.

are placed at the quarter point of a side rather than at the midpoint. This will result in a singularity in strain at the selected special corner.

We first generate a six-node triangle by collapsing the eight-node rectangle shown in Figure 5.16 into a six-node triangle (Figure 5.28). The mapping from the (ξ,η) plane onto the (x,y) plane for the rectangle is

$$x = \sum_{1}^{8} N_i X_i, \quad y = \sum_{1}^{8} N_i Y_i \,, \tag{5.187}$$

where the shape functions are defined by 5.156. Note that the mapping can be used even if the side nodes are not at the mid-point of the side. Now, collapse nodes 4 and 8 onto node 1: $X_4 = X_8 = X_1$ and $Y_4 = Y_8 = Y_1$ and renumber the remaining nodes as in Figure 5.28. We have

$$\begin{aligned} x = &-\frac{1}{2}\xi(1-\xi)X_1 + \frac{1}{4}(1+\xi)(1-\eta)(\xi-\eta-1)X_2 + \frac{1}{4}(1+\xi)(1+\eta)(\xi+\eta-1)X_3 \\ &+\frac{1}{2}(1-\xi^2)(1-\eta)X_4 + \frac{1}{2}(1+\xi)(1-\eta^2)X_4 + \frac{1}{2}(1-\xi^2)(1+\eta)X_6. \end{aligned} \tag{5.188}$$

The formula for y is similar. This induces shape functions for the triangle:

$$\begin{aligned} &N_1 = -\frac{1}{2}\xi(1-\xi), \quad N_2 = \frac{1}{4}(1+\xi)(1-\eta)(\xi-\eta-1), \\ &N_3 = \frac{1}{4}(1+\xi)(1+\eta)(\xi+\eta-1), \quad N_4 = \frac{1}{2}(1-\xi^2)(1-\eta), \\ &N_5 = \frac{1}{2}(1+\xi)(1-\eta^2), \quad N_6 = \frac{1}{2}(1-\xi^2)(1+\eta). \end{aligned} \tag{5.189}$$

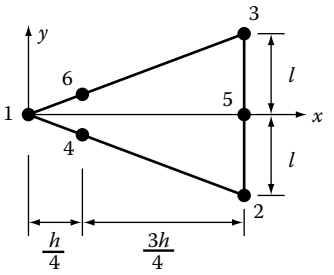


FIGURE 5.28 Singularity triangle.

For the symmetric geometry shown in Figure 5.28, we find

$$x = h(1+\xi)^2/4, \quad y = \frac{l}{4}\eta\left(1+\xi\right)^2. \tag{5.190}$$

The lines for which η is constant are radial lines through node 1. The lines for which ξ is constant are the lines x is constant. The displacements are

$$\begin{aligned} u &= N_1U_1 + N_2U_2 + N_3U_3 + N_4U_4 + N_5U_5 + N_6U_6, \\ v &= N_1V_1 + N_2V_2 + N_3V_3 + N_4V_4 + N_5V_5 + N_6V_6. \end{aligned} \tag{5.191}$$

Calculation of the displacement gradients shows a $1/\sqrt{r}$ singularity, where r is the radial distance from node 1. For example, consider the line $y = 0$, $\eta = 0$, and $x = r$; we then find

$$u = -\frac{1}{4}\left(1-\xi^2\right)\left(2U_1 + U_2 + U_3 - 2U_4 - 2U_6\right) + \frac{1}{2}(1+\xi)U_5 + \frac{1}{2}(1-\xi)U_1. \tag{5.192}$$

Using 5.190, we find the $1/\sqrt{r}$ singularity in the displacement gradient:

$$\frac{\partial u}{\partial x} = c_1 + \frac{c_2}{\sqrt{r}}. \tag{5.193}$$

Similar results are found for the other derivatives. This element is useful for the analysis of structures with crack surfaces.

5.10 MIXED U–P ELEMENTS

For nearly incompressible materials, we can introduce the mean stress as a separate unknown. The appropriate virtual work formulas are 3.113 for three dimensions and 3.21 or 3.24 for plane strain. Shape functions are required for the displacements and for the mean pressure, and they need not be the same functions.

5.10.1 Plane Strain

For plane strain, the virtual work formula is 3.126:

$$\int_{\mathcal{A}} \left\{ 2\mu e_{\alpha\beta}\frac{\partial \overline{u}_\alpha}{\partial x_\beta} - p\frac{\partial \overline{u}_\alpha}{\partial x_\alpha} - \left(\varepsilon_{\alpha\alpha} + \frac{p}{\kappa}\right)\overline{p} \right\} \mathrm{d}A = \int_C T_\alpha \overline{u}_\alpha \,\mathrm{d}s + \int_{\mathcal{A}} b_\alpha \overline{u}_\alpha \mathrm{d}A. \tag{5.194}$$

The displacements and pressure are represented by shape functions

$$\begin{aligned} u &= \sum_k N_k U_k = \mathbf{NU}, \\ v &= \sum_k N_k V_k = \mathbf{NV}, \\ p &= \sum_p H_p P_p = \mathbf{HP}. \end{aligned} \tag{5.195}$$

The shape functions for the mean pressure may involve nodal values shared with adjacent elements or may be expressed entirely in terms of interior nodes or nodeless parameters. In matrix form, **N** and **H** are row matrices. Let us define row matrices:

$$\begin{aligned} A_k &= \frac{\partial N_k}{\partial x}, \quad \mathbf{A} = \frac{\partial \mathbf{N}}{\partial x}, \\ B_k &= \frac{\partial N_k}{\partial y}, \quad \mathbf{B} = \frac{\partial \mathbf{N}}{\partial y}. \end{aligned} \tag{5.196}$$

In matrix form,

$$\mathbf{e} = \begin{bmatrix} e_{11} \\ e_{12} \\ e_{21} \\ e_{22} \end{bmatrix} = \begin{bmatrix} \frac{2}{3}\mathbf{A} & -\frac{1}{3}\mathbf{B} \\ \frac{1}{2}\mathbf{B} & \frac{1}{2}\mathbf{A} \\ \frac{1}{2}\mathbf{B} & \frac{1}{2}\mathbf{A} \\ -\frac{1}{3}\mathbf{A} & \frac{2}{3}\mathbf{B} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}, \quad \varepsilon_{11} + \varepsilon_{22} = \begin{bmatrix} \mathbf{A} & \mathbf{B} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}, \tag{5.197}$$

and

$$\Delta\bar{\mathbf{u}} = \begin{bmatrix} \partial\bar{u}_1/\partial x_1 \\ \partial\bar{u}_1/\partial x_2 \\ \partial\bar{u}_2/\partial x_1 \\ \partial\bar{u}_2/\partial x_2 \end{bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{0} \\ \mathbf{B} & \mathbf{0} \\ \mathbf{0} & \mathbf{A} \\ \mathbf{0} & \mathbf{B} \end{bmatrix} \begin{bmatrix} \bar{\mathbf{U}} \\ \bar{\mathbf{V}} \end{bmatrix}, \quad \partial\bar{u}_\alpha/\partial x_\alpha = \begin{bmatrix} \mathbf{A} & \mathbf{B} \end{bmatrix} \begin{bmatrix} \bar{\mathbf{U}} \\ \bar{\mathbf{V}} \end{bmatrix}. \tag{5.198}$$

In matrix form, the virtual work formula is

$$\int_{\mathcal{R}} \left\{ 2\mu\Delta\bar{\mathbf{u}}^{\mathrm{T}}\mathbf{e} - \partial\bar{u}_\alpha/\partial x_\alpha p - \bar{p}\varepsilon_{\alpha\alpha} - \frac{1}{\kappa}\bar{p}p \right\} \mathrm{d}a = \int_{\mathcal{R}} \bar{\mathbf{u}}^T \mathbf{b}\, \mathrm{d}a + \int_C \bar{\mathbf{u}}^{\mathrm{T}} \mathbf{T}\, \mathrm{d}s. \tag{5.199}$$

Substitution of (5.195)–(5.198) gives

$$\left[\bar{\mathbf{U}}^{\mathrm{T}} \quad \bar{\mathbf{V}}^{\mathrm{T}}\right] \left\{\mathbf{K}_{uu} \begin{bmatrix}\mathbf{U}\\ \mathbf{V}\end{bmatrix} + \mathbf{K}_{up}\mathbf{P}\right\} + \bar{\mathbf{P}}^{\mathrm{T}} \left\{\mathbf{K}_{pu} \begin{bmatrix}\mathbf{U}\\ \mathbf{V}\end{bmatrix} + \mathbf{K}_{pp}\mathbf{P}\right\} \equiv [\bar{\mathbf{U}}^{\mathrm{T}} \quad \bar{\mathbf{V}}^{\mathrm{T}}]\mathbf{F}, \tag{5.200}$$

where

$$\mathbf{K}_{uu} = \int_{\mathcal{R}} 2\mu \begin{bmatrix} \frac{2}{3}\mathbf{A}^{\mathrm{T}}\mathbf{A} + \frac{1}{2}\mathbf{B}^{\mathrm{T}}\mathbf{B} & \frac{1}{2}\mathbf{B}^{\mathrm{T}}\mathbf{A} - \frac{1}{3}\mathbf{A}^{\mathrm{T}}\mathbf{B} \\ \frac{1}{2}\mathbf{A}^{\mathrm{T}}\mathbf{B} - \frac{1}{3}\mathbf{B}^{\mathrm{T}}\mathbf{A} & \frac{2}{3}\mathbf{B}^{\mathrm{T}}\mathbf{B} + \frac{1}{2}\mathbf{A}^{\mathrm{T}}\mathbf{A} \end{bmatrix} \mathrm{d}a, \tag{5.201}$$

$$\mathbf{K}_{pu}^{\mathrm{T}} = \mathbf{K}_{up} = -\int_{\mathcal{R}} \begin{bmatrix}\mathbf{A}^{\mathrm{T}}\\ \mathbf{B}^{\mathrm{T}}\end{bmatrix} \mathbf{H}\, \mathrm{d}a. \tag{5.202}$$

$$\mathbf{K}_{pp} = -\int_{\mathcal{R}} \frac{1}{\kappa}\mathbf{H}^{\mathrm{T}}\mathbf{H}\, \mathrm{d}a. \tag{5.203}$$

Equating the coefficients of the virtual displacements, we find the fundamental relations for the element:

$$\begin{bmatrix}\mathbf{K}_{uu} & \mathbf{K}_{up}\\ \mathbf{K}_{pu} & \mathbf{K}_{pp}\end{bmatrix} \begin{bmatrix}\mathbf{D}\\ \mathbf{P}\end{bmatrix} = \begin{bmatrix}\mathbf{F}\\ \mathbf{0}\end{bmatrix}. \tag{5.204}$$

There are now two possible models. If the interpolation for p uses nodes that are shared with adjacent elements, the element stiffness matrix defined by 5.204 must be merged into the global stiffness matrix. The pressure parameters $\mathbf{P}$ are then determined along with the displacement parameters by solving the global equations.

However, if the interpolation for p uses nodeless parameters or internal nodal parameters, then the pressure parameters $\mathbf{P}$ can be eliminated at the element level by static condensation. Partitioning 5.204 gives

$$\begin{aligned} \mathbf{K}_{uu}\mathbf{D} + \mathbf{K}_{up}\mathbf{P} &= \mathbf{F}, \\ \mathbf{K}_{pu}\mathbf{D} + \mathbf{K}_{pp}\mathbf{P} &= \mathbf{0}. \end{aligned} \tag{5.205}$$

Eliminating **P** gives

$$\mathbf{KD} = \mathbf{F}, \tag{5.206}$$

where

$$\mathbf{K} = \mathbf{K}_{uu} - \mathbf{K}_{up}\mathbf{K}_{pp}^{-1}\mathbf{K}_{pu}. \tag{5.207}$$

The analysis then proceeds as in the pure displacement model but using the stiffness matrix defined by 5.207.

In practice, if the material is nearly incompressible the model is typically too stiff and may "lock" unless the interpolation or the integration of the pressure terms is of lower order than for the displacement terms. The simplest usable case is the following four-node element. The displacements are interpolated using the serendipity functions 5.10 that are linear in (x,y). The pressure is interpolated by one order lower, that is, by a constant pressure in each element. In this case, we have the following results.

$$\mathbf{A} = \left[\frac{1}{2a}\xi_k(1+\eta_k\eta)\right], \quad \mathbf{B} = \left[\frac{1}{2b}\eta_k(1+\xi_k\xi)\right], \quad \mathbf{H} = [1]. \tag{5.208}$$

For the rectangular element, the integration can be easily done. We find the stiffness matrix **K** defined by 5.207 to be as follows.

$$\mathbf{K} = \begin{bmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} \\ \mathbf{K}_{21} & \mathbf{K}_{22} \end{bmatrix}, \tag{5.209}$$

where each submatrix is 4×4 with elements:

$$\mathbf{K}_{11} = \left[\frac{\mu}{36}\left(4\frac{b}{a}\xi_k\xi_m(3+\eta_k\eta_m) + 3\frac{a}{b}\eta_k\eta_m(3+\xi_k\xi_m)\right) + \frac{\kappa}{4}\frac{b}{a}\xi_k\xi_m\right], \tag{5.210}$$

$$\mathbf{K}_{22} = \left[\frac{\mu}{36}\left(4\frac{a}{b}\eta_k\eta_m(3+\xi_k\xi_m) + 3\frac{b}{a}\xi_k\xi_m(3+\eta_k\eta_m)\right) + \frac{\kappa}{4}\frac{a}{b}\eta_k\eta_m\right], \tag{5.211}$$

$$\mathbf{K}_{12}^{\mathrm{T}} = \mathbf{K}_{21} = \left[\frac{\mu}{12}(3\xi_k\eta_m - 2\eta_k\xi_m) + \frac{\kappa}{4}\eta_k\xi_m\right]. \tag{5.212}$$

In the case where $a = b$, $\nu = 1/3$, $\kappa = 8\mu/3$,

$$\mathbf{K}_{11} = \frac{\mu}{36}\begin{bmatrix} 52 & -34 & -44 & 26 \\ -34 & 52 & 26 & -44 \\ -44 & 26 & 52 & -34 \\ 26 & -44 & -34 & 52 \end{bmatrix}, \tag{5.213}$$

$$\mathbf{K}_{21} = \frac{\mu}{36}\begin{bmatrix} 27 & -9 & -27 & 9 \\ 9 & -27 & -9 & 27 \\ -27 & 9 & 27 & -9 \\ -9 & 27 & 9 & -27 \end{bmatrix}, \tag{5.214}$$

and

$$\mathbf{K}_{22} = \frac{\mu}{36}\begin{bmatrix} 52 & 20 & -38 & -34 \\ 20 & 52 & -34 & -38 \\ -38 & -34 & 52 & 20 \\ -34 & -38 & 20 & 52 \end{bmatrix}. \tag{5.215}$$

The first row of **K** is

$$\begin{matrix} & U_1 & U_2 & U_3 & U_4 & V_1 & V_2 & V_3 & V_4 \end{matrix}$$
$$[K_{1i}] = \frac{\mu}{36}\begin{bmatrix} 52 & -34 & -38 & 20 & 27 & 9 & -27 & -9 \end{bmatrix}. \tag{5.216}$$

This can be compared with the serendipity element 5.29, which, in this case of plane strain, has the first row

$$[K_{1i}] = \frac{\mu}{36}\begin{bmatrix} 60 & -42 & -30 & 12 & 27 & 9 & -27 & -9 \end{bmatrix}. \tag{5.217}$$

5.10.2 Alternative Formulation for Plane Strain

The alternative form (3.129) of the virtual work formula for plain strain,

$$\int_{\mathcal{A}} \left\{ 2\mu\varepsilon_{\alpha\beta}\frac{\partial \bar{u}_\alpha}{\partial x_\beta} - \beta p \frac{\partial \bar{u}_\alpha}{\partial x_\alpha} - \left(\varepsilon_{\alpha\alpha} + \frac{p}{\kappa}\right)\beta \bar{p} \right\} \mathrm{d}A = \int_C T_\alpha \bar{u}_\alpha \,\mathrm{d}s + \int_{\mathcal{A}} b_\alpha \bar{u}_\alpha \,\mathrm{d}A,$$
$$\beta = \frac{\lambda}{\kappa}, \tag{5.218}$$

leads to a slightly different stiffness matrix. From 5.195 and 5.196, we have

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{12} \\ \varepsilon_{21} \\ \varepsilon_{22} \end{bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{0} \\ \frac{1}{2}\mathbf{B} & \frac{1}{2}\mathbf{A} \\ \frac{1}{2}\mathbf{B} & \frac{1}{2}\mathbf{A} \\ \mathbf{0} & \mathbf{B} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \end{bmatrix}. \tag{5.219}$$

In place of 5.201–5.203, we obtain

$$\mathbf{K}_{uu} = \int_{\mathcal{R}} \mu \begin{bmatrix} 2\mathbf{A}^{\mathrm{T}}\mathbf{A} + \mathbf{B}^{\mathrm{T}}\mathbf{B} & \mathbf{B}^{\mathrm{T}}\mathbf{A} \\ \mathbf{A}^{\mathrm{T}}\mathbf{B} & 2\mathbf{B}^{\mathrm{T}}\mathbf{B} + \mathbf{A}^{\mathrm{T}}\mathbf{A} \end{bmatrix} \mathrm{d}a, \tag{5.220}$$

$$\mathbf{K}_{pu}^{\mathrm{T}} = \mathbf{K}_{up} = -\int_{\mathcal{R}} \beta \begin{bmatrix} \mathbf{A}^{\mathrm{T}} \\ \mathbf{B}^{\mathrm{T}} \end{bmatrix} \mathbf{H}\, \mathrm{d}a, \tag{5.221}$$

$$\mathbf{K}_{pp} = -\int_{\mathcal{R}} \frac{\beta}{\kappa} \mathbf{H}^{\mathrm{T}}\mathbf{H}\, \mathrm{d}a. \tag{5.222}$$

For the serendipity element with constant pressure, the stiffness matrix is again calculated by 5.207. Using 5.208,

$$-\mathbf{K}_{up}\mathbf{K}_{pp}^{-1}\mathbf{K}_{pu} = ab\lambda \begin{bmatrix} \mathbf{A}(0)^{\mathrm{T}}\mathbf{A}(0) & \mathbf{A}(0)^{\mathrm{T}}\mathbf{B}(0) \\ \mathbf{B}(0)^{\mathrm{T}}\mathbf{A}(0) & \mathbf{B}(0)^{\mathrm{T}}\mathbf{B}(0) \end{bmatrix} \tag{5.223}$$

since

$$\iint A_k\, \mathrm{d}x\, \mathrm{d}y = \frac{ab}{4} \int_{-1}^{1}\int_{-1}^{1} A_k\, \mathrm{d}\xi\, \mathrm{d}\eta = \frac{b}{2}\xi_k = abA_k(0)\,. \tag{5.224}$$

The complete stiffness matrix is the sum of 5.220 and 5.223, and is exactly the same as matrix 5.98 for selective reduced integration.

In the case where $a = b$, $\nu = 1/3$, $\kappa = 8\mu/3$, the first column of $\mathbf{K}$ is

$$[K_{i1}]^{\mathrm{T}} = \frac{\mu}{36}\begin{bmatrix} 54 & -36 & -36 & 18 & 27 & 9 & -27 & -9 \end{bmatrix}. \tag{5.225}$$

This agrees with ANSYS element Plane182 mixed U/P method with the Enhanced Strain option.

5.10.3 3D Elements

The modified virtual work formula 3.110 is

$$\int_{\mathbf{V}} \left\{ \left(2\mu e_{ij} - p\delta_{ij}\right)\overline{\varepsilon}_{ij} - \left(\varepsilon + \frac{p}{\kappa}\right)\overline{p} \right\} \mathrm{d}V = \int_{\mathbf{A}} T_i \overline{u}_i \, \mathrm{d}A + \int_{\mathbf{V}} b_i \overline{u}_i \, \mathrm{d}V, \tag{5.226}$$

The displacements are represented by shape functions

$$\begin{aligned} u &= \sum_k N_k U_k = \mathbf{NU}, \quad v = \sum_k N_k V_k = \mathbf{NV}, \\ w &= \sum_k N_k W_k = \mathbf{NW}, \quad p = \sum_p H_p P_p = \mathbf{HP}. \end{aligned} \tag{5.227}$$

We define row matrices:

$$\begin{aligned} A_k &= \frac{\partial N_k}{\partial x}, \quad \mathbf{A} = \frac{\partial \mathbf{N}}{\partial x}, \\ B_k &= \frac{\partial N_k}{\partial y}, \quad \mathbf{B} = \frac{\partial \mathbf{N}}{\partial y}, \\ C_k &= \frac{\partial N_k}{\partial y}, \quad \mathbf{C} = \frac{\partial \mathbf{N}}{\partial y}. \end{aligned} \tag{5.228}$$

The deviatoric strains and volumetric strain are then

$$\begin{bmatrix} e_{11} \\ e_{22} \\ e_{33} \\ e_{12} \\ e_{23} \\ e_{31} \end{bmatrix} = \begin{bmatrix} \frac{2}{3}\mathbf{A} & -\frac{1}{3}\mathbf{B} & -\frac{1}{3}\mathbf{C} \\ -\frac{1}{3}\mathbf{A} & \frac{2}{3}\mathbf{B} & -\frac{1}{3}\mathbf{C} \\ -\frac{1}{3}\mathbf{A} & -\frac{1}{3}\mathbf{B} & \frac{2}{3}\mathbf{C} \\ \frac{1}{2}\mathbf{B} & \frac{1}{2}\mathbf{A} & \mathbf{0} \\ \mathbf{0} & \frac{1}{2}\mathbf{C} & \frac{1}{2}\mathbf{B} \\ \frac{1}{2}\mathbf{C} & \mathbf{0} & \frac{1}{2}\mathbf{A} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \\ \mathbf{W} \end{bmatrix}, \quad \boldsymbol{\varepsilon}_{ii} = \begin{bmatrix} \mathbf{A} & \mathbf{B} & \mathbf{C} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \\ \mathbf{W} \end{bmatrix}. \tag{5.229}$$

The virtual strains are

$$\begin{bmatrix} \bar{\varepsilon}_{11} \\ \bar{\varepsilon}_{22} \\ \bar{\varepsilon}_{33} \\ 2\bar{\varepsilon}_{12} \\ 2\bar{\varepsilon}_{23} \\ 2\bar{\varepsilon}_{31} \end{bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{B} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{C} \\ \mathbf{B} & \mathbf{A} & \mathbf{0} \\ \mathbf{0} & \mathbf{C} & \mathbf{B} \\ \mathbf{C} & \mathbf{0} & \mathbf{A} \end{bmatrix} \begin{bmatrix} \bar{\mathbf{U}} \\ \bar{\mathbf{V}} \\ \bar{\mathbf{W}} \end{bmatrix}. \tag{5.230}$$

Substituting into the virtual work formula and equating the coefficients of the virtual parameters gives the governing equations:

$$\begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{up} \\ \mathbf{K}_{pu} & \mathbf{K}_{pp} \end{bmatrix} \begin{bmatrix} \mathbf{U} \\ \mathbf{V} \\ \mathbf{W} \\ \mathbf{P} \end{bmatrix} = \begin{bmatrix} \mathbf{F} \\ \mathbf{0} \end{bmatrix}, \tag{5.231}$$

where

$$\mathbf{K}_{uu} = \int_{\mathcal{V}} 2\mu \begin{bmatrix} \frac{2}{3}\mathbf{A}^{\mathrm{T}}\mathbf{A} + \frac{1}{2}\mathbf{B}^{\mathrm{T}}\mathbf{B} + \frac{1}{2}\mathbf{C}^{\mathrm{T}}\mathbf{C} & \frac{1}{2}\mathbf{B}^{\mathrm{T}}\mathbf{A} - \frac{1}{3}\mathbf{A}^{\mathrm{T}}\mathbf{B} & \frac{1}{2}\mathbf{C}^{\mathrm{T}}\mathbf{A} - \frac{1}{3}\mathbf{A}^{\mathrm{T}}\mathbf{C} \\ \frac{1}{2}\mathbf{A}^{\mathrm{T}}\mathbf{B} - \frac{1}{3}\mathbf{B}^{\mathrm{T}}\mathbf{A} & \frac{2}{3}\mathbf{B}^{\mathrm{T}}\mathbf{B} + \frac{1}{2}\mathbf{A}^{\mathrm{T}}\mathbf{A} + \frac{1}{2}\mathbf{C}^{\mathrm{T}}\mathbf{C} & \frac{1}{2}\mathbf{C}^{\mathrm{T}}\mathbf{B} - \frac{1}{3}\mathbf{B}^{\mathrm{T}}\mathbf{C} \\ \frac{1}{2}\mathbf{A}^{\mathrm{T}}\mathbf{C} - \frac{1}{3}\mathbf{C}^{\mathrm{T}}\mathbf{A} & \frac{1}{2}\mathbf{B}^{\mathrm{T}}\mathbf{C} - \frac{1}{3}\mathbf{C}^{\mathrm{T}}\mathbf{B} & \frac{2}{3}\mathbf{C}^{\mathrm{T}}\mathbf{C} + \frac{1}{2}\mathbf{B}^{\mathrm{T}}\mathbf{B} + \frac{1}{2}\mathbf{A}^{\mathrm{T}}\mathbf{A} \end{bmatrix} \mathrm{d}V, \tag{5.232}$$

$$\mathbf{K}_{pu}^{\mathrm{T}} = \mathbf{K}_{up} = -\int_{\mathcal{V}} \begin{bmatrix} \mathbf{A}^{\mathrm{T}} \\ \mathbf{B}^{\mathrm{T}} \\ \mathbf{C}^{\mathrm{T}} \end{bmatrix} \mathbf{H} \mathrm{d}V, \tag{5.233}$$

$$\mathbf{K}_{pp} = -\int_{\mathcal{V}} \frac{1}{\kappa} \mathbf{H}^{\mathrm{T}}\mathbf{H} \, \mathrm{d}V \tag{5.234}$$

As an example, we consider the eight-node brick using the serendipity functions for the displacement and a constant pressure in each element. We will use centered coordinates:

$$-\frac{a}{2} \le x \le \frac{a}{2}, \quad -\frac{b}{2} \le y \le \frac{b}{2}, \quad -\frac{c}{2} \le z \le \frac{c}{2}. \tag{5.235}$$

The serendipity shape functions for the displacements are

$$N_i = \frac{1}{8}(1+\xi_i\xi)(1+\eta_i\eta)(1+\zeta_i\zeta). \tag{5.236}$$

Therefore,

$$\begin{aligned}
\mathbf{A} &= \frac{2}{a}\left\lfloor \partial N_k/\partial\xi \right\rfloor = \frac{2}{a}\left\lfloor \xi_k(1+\eta_k\eta)(1+\zeta_k\zeta) \right\rfloor,\\
\mathbf{B} &= \frac{2}{b}\left\lfloor \partial N_k/\partial\eta \right\rfloor = \frac{2}{b}\left\lfloor \eta_k(1+\zeta_k\zeta)(1+\xi_k\xi) \right\rfloor,\\
\mathbf{C} &= \frac{2}{c}\left\lfloor \partial N_k/\partial\zeta \right\rfloor = \frac{2}{c}\left\lfloor \zeta_k(1+\xi_k\xi)(1+\eta_k\eta) \right\rfloor.
\end{aligned} \tag{5.237}$$

Therefore,

$$\begin{aligned}
\int_{\mathcal{V}} \mathbf{A}^{\mathrm{T}}\mathbf{A}\,\mathrm{d}V &= \frac{abc}{144a^2}\left\lfloor \xi_k\xi_m(3+\eta_k\eta_m)(3+\zeta_k\zeta_m) \right\rfloor,\\
\int_{\mathcal{V}} \mathbf{A}^{\mathrm{T}}\mathbf{B}\,\mathrm{d}V &= \frac{abc}{48ab}\left\lfloor \xi_k\eta_m(3+\zeta_k\zeta_m) \right\rfloor.
\end{aligned} \tag{5.238}$$

The other factors can be written down by cyclic permutation of (ξ,η,ζ) and (a,b,c). For constant pressure in each element, $\mathbf{H} = [1]$.

For the case $a = b = c = 2$, $\nu = 1/3$, $\kappa = 8\mu/3$, the first column of the element stiffness matrix is

$$[FX_k] = \frac{\mu}{27}\begin{bmatrix} 29 \\ 10 \\ 5 \\ 10 \\ -11 \\ -14.5 \\ -14 \\ -14.5 \end{bmatrix}, \quad [FY_k] = \frac{\mu}{27}\begin{bmatrix} 10.5 \\ -1.5 \\ -5.25 \\ 9.75 \\ 1.5 \\ -10.5 \\ -9.75 \\ 5.25 \end{bmatrix}, \quad [FZ_k] = \frac{\mu}{27}\begin{bmatrix} 10.5 \\ 9.75 \\ -5.25 \\ -1.5 \\ 1.5 \\ 5.25 \\ -9.75 \\ -10.5 \end{bmatrix}. \tag{5.239}$$

This can be compared with ANSYS element SOLID185, U–P method, Full Integration.

5.11 PROBLEMS

1. Given: The stiffness matrix for a plane stress square element of Figure 5.29 is given by

$$\mathbf{k} = \begin{bmatrix} 6 & 1.5 & -3 & -1.5 & -3 & -1.5 & 0 & 1.5 \\ & 6 & 1.5 & 0 & -1.5 & -3 & -1.5 & -3 \\ & & 6 & -1.5 & 0 & -1.5 & -3 & 1.5 \\ & & & 6 & 1.5 & -3 & 1.5 & -3 \\ & & & & 6 & 1.5 & -3 & -1.5 \\ & & & & & 6 & 1.5 & 0 \\ & & & & & & 6 & -1.5 \\ & & & & & & & 6 \end{bmatrix} \begin{matrix} U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \\ U_4 \\ V_4 \end{matrix}. \tag{5.240}$$

A plane stress problem is modeled by two elements as shown in Figure 5.30. Determine the global stiffness matrix **K** for the unrestrained body.
Answer: Stiffness matrix for element 1 with nodes 1–2–5–6:

$$\mathbf{k}^1 = \begin{matrix} D_1 & D_2 & D_3 & D_4 & D_9 & D_{10} & D_{11} & D_{12} \end{matrix} \\ \begin{bmatrix} 6 & 1.5 & -3 & -1.5 & -3 & -1.5 & 0 & 1.5 \\ & 6 & 1.5 & 0 & -1.5 & -3 & -1.5 & -3 \\ & & 6 & -1.5 & 0 & -1.5 & -3 & 1.5 \\ & & & 6 & 1.5 & -3 & 1.5 & -3 \\ & & & & 6 & 1.5 & -3 & -1.5 \\ & & & & & 6 & 1.5 & 0 \\ & & & & & & 6 & -1.5 \\ & & & & & & & 6 \end{bmatrix} \begin{matrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_9 \\ F_{10} \\ F_{11} \\ F_{12} \end{matrix}.$$

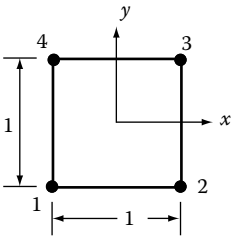


FIGURE 5.29 Four-node element.

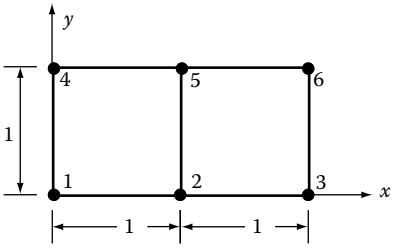


FIGURE 5.30 Two-element model (a).

Stiffness matrix for element 2 with nodes 2–3–4–5:

$$\mathbf{k}^2 = \begin{array}{c} \begin{matrix} D_3 & D_4 & D_5 & D_6 & D_7 & D_8 & D_9 & D_{10} \end{matrix} \\ \begin{bmatrix} 6 & 1.5 & -3 & -1.5 & -3 & -1.5 & 0 & 1.5 \\ & 6 & 1.5 & 0 & -1.5 & -3 & -1.5 & -3 \\ & & 6 & -1.5 & 0 & -1.5 & -3 & 1.5 \\ & & & 6 & 1.5 & -3 & 1.5 & -3 \\ & & & & 6 & 1.5 & -3 & -1.5 \\ & & & & & 6 & 1.5 & 0 \\ & & & & & & 6 & -1.5 \\ & & & & & & & 6 \end{bmatrix} \begin{matrix} F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \\ F_9 \\ F_{10} \end{matrix} \end{array}.$$

Global stiffness matrix after merging element 1:

$$\mathbf{K} = \begin{array}{c} \begin{matrix} D_1 & D_2 & D_3 & D_4 & D_5 & D_6 & D_7 & D_8 & D_9 & D_{10} & D_{11} & D_{12} \end{matrix} \\ \begin{bmatrix} 6 & 1.5 & -3 & -1.5 & & & & & -3 & -1.5 & 0 & 1.5 \\ & 6 & 1.5 & 0 & & & & & -1.5 & -3 & -1.5 & -3 \\ & & 6 & -1.5 & & & & & 0 & -1.5 & -3 & 1.5 \\ & & & 6 & & & & & 1.5 & -3 & 1.5 & -3 \\ & & & & & & & & & & & \\ & & & & & & & & & & & \\ & & & & & & & & & & & \\ & & & & & & & & & & & \\ & & & & & & & & 6 & 1.5 & -3 & -1.5 \\ & & & & & & & & & 6 & 1.5 & 0 \\ & & & & & & & & & & 6 & -1.5 \\ & & & & & & & & & & & 6 \end{bmatrix} \begin{matrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \\ F_9 \\ F_{10} \\ F_{11} \\ F_{12} \end{matrix} \end{array}.$$

Global stiffness matrix after merging both elements:

$$\mathbf{K} = \begin{array}{c} \begin{matrix} D_1 & D_2 & D_3 & D_4 & D_5 & D_6 & D_7 & D_8 & D_9 & D_{10} & D_{11} & D_{12} \end{matrix} \\ \begin{bmatrix} 6 & 1.5 & -3 & -1.5 & 0 & 0 & 0 & 0 & -3 & -1.5 & 0 & 1.5 \\ & 6 & 1.5 & 0 & 0 & 0 & 0 & 0 & -1.5 & -3 & -1.5 & -3 \\ & & 12 & 0 & -3 & -1.5 & -3 & -1.5 & 0 & 0 & -3 & 1.5 \\ & & & 12 & 1.5 & 0 & -1.5 & -3 & 0 & -6 & 1.5 & -3 \\ & & & & 6 & -1.5 & 0 & -1.5 & -3 & 1.5 & 0 & 0 \\ & & & & & 6 & 1.5 & -3 & 1.5 & -3 & 0 & 0 \\ & & & & & & 6 & 1.5 & -3 & -1.5 & 0 & 0 \\ & & & & & & & 6 & 1.5 & 0 & 0 & 0 \\ & & & & & & & & 12 & 0 & -3 & -1.5 \\ & & & & & & & & & 12 & 1.5 & 0 \\ & & & & & & & & & & 6 & -1.5 \\ & & & & & & & & & & & 6 \end{bmatrix} \end{array} \begin{matrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \\ F_9 \\ F_{10} \\ F_{11} \\ F_{12} \end{matrix}.$$

2. Given: The stiffness matrix for a plane stress square element of Figure 5.29 is given by Equation 5.240. A plane stress problem is modeled by two elements as shown in Figure 5.31.

   Determine: (a) the global stiffness matrix **K** for the unrestrained body; (b) the reduced stiffness matrix that must be inverted to obtain the nodal displacements if nodes 1, 3, 4, and 6 have zero displacement in the x and y directions.
3. Determine the element nodal forces for a constant traction $T_2 = p$ on side 2–3 of the four-node element (Figure 5.1) by using Equation 3.16.
4. Determine the global force matrix **F** for a constant traction $T_1 = p$ on the side of the plane region if the region is meshed by the four node elements as shown in Figure 5.32.

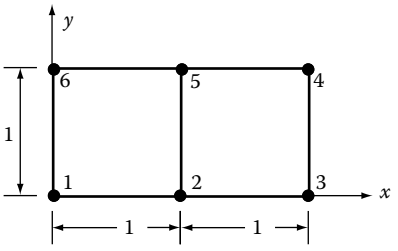


FIGURE 5.31 Two-element model (b).

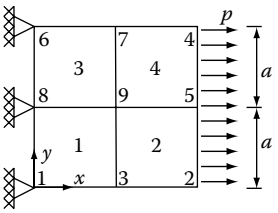


FIGURE 5.32 Four-element model.

5. Solve the plane stress problem diagrammed in Figure 5.33 for the corner displacement using one 4-node square element numbered as shown in Figure 5.29 without using ANSYS.

   Answers:

$$\mathbf{K} = \frac{\mu}{24}\begin{bmatrix} 32 & & & & & & & \\ 12 & 32 & & & & & & \\ -20 & 0 & 32 & & & & & \\ 0 & 4 & -12 & 32 & & & & \\ -16 & -12 & 4 & 0 & 32 & & & \\ -12 & -16 & 0 & -20 & 12 & 32 & & \\ 4 & 0 & -16 & 12 & -20 & 0 & 32 & \\ 0 & -20 & 12 & -16 & 0 & 4 & -12 & 32 \end{bmatrix},$$

$$U = \frac{14}{3}\frac{pa}{E}, \quad V = 2\frac{pa}{E}.$$

6. Use ANSYS to solve the plane stress problem shown in Figure 5.33 for $a = 1$, $p = 1$, $E = 100$, $\nu = 0.3$, using 16 equal-size four-node rectangles. Input the nodal forces by hand. Submit a list of nodal displacements. (See Section 15.7.)

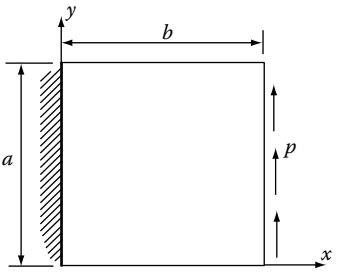


FIGURE 5.33 Short beam problem.

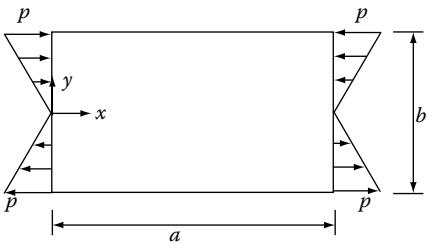


FIGURE 5.34 Plane stress problem.

7. Derive the generalized nodal forces for a uniform body force $b_1 = 1$ over a four-node rectangular element in plane stress.
8. Consider the plane stress problem shown in Figure 5.34.

   Given: $a = 10$, $b = 2$, $p = 1$, $E = 1$, $\nu = 0.3$. The origin has zero displacement and $\nu = 0$ at point $(a,0)$. (a) Solve this problem using ANSYS with element Plane182 and square elements of dimension 1/2. (b) Compare the displacement UY and the stress SX along line $x = 5$ with the exact solution discussed in Section 2.1.11. (See Section 15.9.)
9. Solve problem 8 using the eight-node element Plane183. Submit the nodal displacements and nodal stress along line $x = 5$, and compare the result with the exact solution.
10. Given the shape functions for the four-node rectangle (5.10), obtain the shape functions for a right triangle by setting $X_1 = X_4$, $Y_1 = Y_4$, $U_1 = U_4$, and $V_1 = V_4$. Show that these shape functions agree with those for the three-node triangle if $\xi = 2\zeta_2 + 2\zeta_3 - 1$ and $\eta = (\zeta_3 - \zeta_2)/(\zeta_3 + \zeta_2)$.
11. Determine the nodal forces for $T_1 =$ constant, on the side $\xi = 1$ of the eight-node rectangle (Figure 5.16).
12. Solve the plane strain bending problem (Section 5.1.3) using the Plane183 element and a 4×8 mesh for $c = 2$, $L = 16$, i = 8192, $\nu = 0.3$. Compare the displacement with that for Plane182 elements (Table 5.2) and with the exact result. (See Section 15.10.)
13. Solve the plane strain bending problem (Section 5.1.3) using the Plane182 element with the U–P formulation (OPTIONS > K8 = MIXED U/P) and a

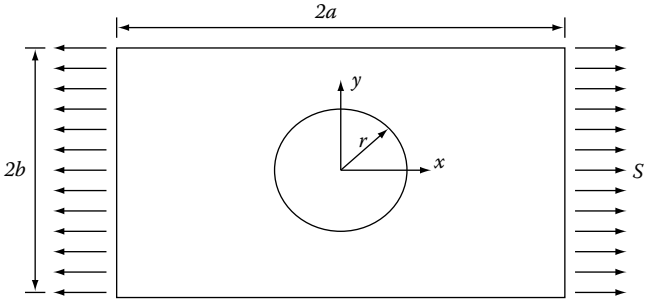


FIGURE 5.35 Sheet with a hole.

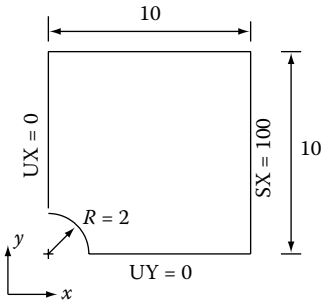


FIGURE 5.36 Plane stress: sheet with a hole

4×8 mesh for $c = 2$, $L = 16$, $E = 8192$, $\nu = 0.499$. Compare the y displacement with that for Plane42 elements (Table 5.2) and with the exact result.

14. Solve problem 6 by ANSYS using SURF153 to apply the distributed load. (See Section 15.11.)
15. Show that the displacement along an edge of the eight-node rectangle varies quadratically and is therefore uniquely determined by the values at the three nodes on the edge. (This ensures continuity between elements by matching nodal displacements.)
16. Determine shape functions for a triangle with mid-side nodes from the eight-node quadrilateral by collapsing nodes 1, 4, and 8 together.
17. A sheet with a central hole (see Figure 5.35) is stretched by a uniform edge stress S resulting in a stress concentration at the hole. In nondimensional variables: $S = 100$, $a = 20$, $b = 10$, $r = 5$, $E = 1$, $\nu = 0.3$. Use ANSYS with element Plane182 to investigate the stress concentration (see Section 15.12). Use 1/4 of the sheet and symmetry conditions (Figure 5.36).
18. Solve problem 17 using element Plane183. Plot stress SX along $x = 0$.
19. Evaluate $\int_{\mathbf{A}} \mathbf{A}_2^{\mathrm{T}} \mathbf{A}_1 \, \mathrm{d}a$ for the four-node rectangle (Section 5.1).
20. For the reduced integration on the λ term, evaluate fully the λ part of the element stiffness matrix: $\mathbf{k}_2 = ab\mathbf{A}(0,0)^{\mathrm{T}}\mathbf{C}_2\mathbf{A}(0,0)$.
21. Calculate all components of the consistent element force matrix for a uniformly distributed load p in the x direction on side $\xi = 1$ of the eight-node rectangle.
22. Determine matrix $\mathbf{B}$ for the Wilson–Taylor four-node rectangle (see Section 5.2.1).

REFERENCES

1. Wilson, E. L. et al., Incompatible displacement models, *Numerical and Computer Methods in Structural Mechanics*, ed. S. J. Fenves et al., 43–57, Academic Press, New York, 1973.
2. Taylor, R. L., P. J. Beresford, and E. L. Wilson, A non-conforming element for stress analysis, *Int. J. Numer. Meth. Eng.,* 10, 1211–1219, 1976.

3. Simo, J. C., and M. S. Rifai, A class of mixed assumed strain methods and the method of incompatible modes, *Int. J. Numer. Meth. Eng.*, 29, 1595–1638, 1990.
4. Hughes, T. J. R., Generalization of selective integration procedures to anisotropic and nonlinear media, *Int. J. Numer. Meth. Eng*., 15, 1413–1418, 1980.
5. Kosloff, D., and G. A. Frazier, Treatment of hourglass patterns in low order finite element codes, *Int. J. Numer. Anal. Meth. Geomech.*, 2, 57–72, 1978.
6. Flanagan, D. P., and T. Belytschko, A uniform strain hexahedron and quadrilateral with orthogonal hourglass control, *Int. J. Numer. Meth. Eng.*, 17, 679–706, 1981.
7. Kardestuncer, H., and D. H. Norrie, eds., *Finite Element Handbook*, 2.103, McGraw-Hill, New York, 1987.
8. Irons, B. M., Engineering applications of numerical integration in stiffness methods, *AIAA J.*, 4, 2035–2037, 1966.
9. Kardestuncer, H., and D. H. Norrie, eds., *Finite Element Handbook*, 2.121–2.131, McGraw-Hill, New York, 1987.

BIBLIOGRAPHY

Bathe, K. J., *Finite Element Procedures*, Prentice-Hall, Englewood Cliffs, NJ, 1996.

Hughes, T. J. R., *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*, Dover Publications, Mineola, NY, 2000.

Zienkiewicz, O. C., R. L. Taylor, and J. Z. Zhu, *The Finite Element Method: Its Basis and Fundamentals*, Elsevier Butterworth-Heinemann, Burlington, MA, 6th ed., 2005.

Zienkiewicz, O. C., and R. L. Taylor, *The Finite Element Method for Solid and Structural Mechanics*, Elsevier Butterworth-Heinemann, Burlington, MA, 6th ed., 2005.

6 Errors and Convergence of Finite Element Solution

6.1 GENERAL REMARKS

From the engineer's viewpoint, the objective in the finite element method is to establish an approximate solution to the boundary value problem that is sufficiently accurate for engineering design. That is, one hopes that the error can be made reasonably small by an analysis using the number of elements that can be handled by the available computer. From the mathematician's viewpoint, the objective is to establish a numerical procedure that ensures that any prescribed measure of error can be made as small as desired by taking enough elements, even if such calculations cannot actually be performed, that is, the mathematician seeks proof of convergence. The engineer seeks to achieve small error within the available capacity of the computer.

Error in the calculation has several sources. First of these is the simple inaccuracy of all real numerical calculations. Each machine is capable of only a finite number of digits in the representation of real numbers. Every multiplication or division therefore introduces a round-off error. Such errors can be large if the stiffness matrix is poorly conditioned. This can happen, for example, when one portion of the body is very stiff compared with the remainder. In this case, too many nodes are being used to describe what is essentially a rigid motion.

Another source of error is approximation of boundary conditions. The conditions on the trial functions for minimizing the potential may be violated because the finite element approximation fails to exactly satisfy the boundary condition on displacement at every point. The element boundaries may not exactly match the actual boundary. Even when the shapes match, the displacements are specified only at the node points and the element may be such that the displacement boundary conditions are not exactly satisfied at all points between the nodes. This kind of error does not generally prohibit convergence, provided that the boundaries do match in the limit of vanishing element size.

Numerical integration will also introduce an error. It is usually a compensating error because numerical integration effectively makes the model more flexible. Using a number of integration points that are less than that required for an exact integration over the region can yield a better answer than is obtained by exact integration of the energy over the element. Convergence still occurs if this error vanishes with decreasing element size.

The error in calculation of the displacement or stress is, of course, greatest where those quantities vary most rapidly. For the best results with the same number of elements, one should not use a uniform element size. Smaller elements should be used where the stress is varying rapidly. As shown in the short beam example in Section 5.1.4, smaller elements should be used near the corners. The ideal element distribution is one in which

the error in the strain energy of an element is the same for all elements. This can be achieved by making a sequence of calculations, beginning with a uniform coarse mesh of elements, and placing more elements in regions where high displacement gradients are found. In this way, the mesh is adapted to suit the problem. There are, in fact, several computer programs that offer adaptive mesh generation.

Proof of convergence is a complicated mathematical problem in the theory of numerical analysis.[1] Roughly, convergence will occur if the finite element model generates a continuous displacement field (continuity requirement), and the linear terms in a power series representation of displacement within an element are present (completeness requirement). Convergence to the true solution may occur in other cases but this must be separately established.

Theoretical studies of converge only establish that the error in displacements vanishes with the element size. The actual magnitude of the error for any finite number of elements cannot be established. As a practical matter, the magnitude of the error is estimated by doing a series of calculations with increasingly larger numbers of elements and plotting the results to see if there is an apparent limit. However, care must be taken in the interpretation of the stress calculation. Singularities can occur so that the calculated stress does not approach a limit, but just becomes larger and larger for each element subdivision without a limiting value. We have seen that such a singularity exists in the corner for the short beam as discussed in Section 2.1.14.

Such calculations for the displacement of the short beam example using the triangular element and the four-node rectangle are plotted in Figure 4.9 and Figure 5.25. The displacement does appear to be monotonically converging to a definite limit that we take to be the true solution. There is no guarantee, however, that the limit will be evident with any finite number of elements. We are forced to rely on experience and comparison to the exact solution in a few cases where the exact solution is known. In practice, one must be satisfied when the incremental improvement in displacement is a small fraction of the last calculated value. It is also possible to extrapolate the solution as described in Section 4.4.1.

Since the true solution makes the potential energy a minimum, the finite element approximation gives a larger potential energy and, in the sense that has been described in Section 3.1.5, the calculated displacements are always too small. If the elements are chosen appropriately, the numerical solution coincides with the true solution in the limit of vanishing element size. In the process of choosing increasing numbers of elements, if each successive subdivision includes the previous one as a special case, the potential energy will become monotonically smaller, and the calculated displacements will become monotonically larger, that is, the finite element solution will exhibit monotonic convergence from below. This is the situation for the analysis illustrated in Figure 5.25.

In the derivation of the potential energy, it has been assumed that displacement components are continuous functions of space variables and the strains derived from displacement are continuous, except possibly for a finite number of simple jump discontinuities. If the finite element model has this property, the elements are said to be conforming or compatible. Nonconforming elements violate the continuity requirement at element boundaries. Cases are known where convergence occurs even for a nonconforming element, but in other cases the solution does not converge, or converges to a function that is not the true solution.

Nevertheless, nonconforming elements are sometimes used. In some cases, the nonconforming elements are used to obtain a more reasonable approximation to the actual state of stress. In other cases, such as for plates and shells, they may be an attempt to simplify the formulation when it is difficult to use shape functions for conforming elements, or none have been found, or the conforming elements lead to numerical difficulties.

Nonconforming elements often yield better answers for a small number of elements than the more theoretically sound conforming elements. The conforming element provides a model of the structure that is always too stiff. The nonconforming element provides a means of relaxing the excessive constraints of the conforming element and therefore may achieve a better approximation—that is, the nonconforming element introduces a compensating error. Monotonic convergence may be lost as a result of relaxing the constraints, and the approximation provided by the nonconforming element may sometimes give displacements that are too large.

Caution is necessary in using nonconforming elements. In the literature, some elements have been proposed that do not provide convergence, or for which convergence is to the wrong answer, or for which convergence depends on the shape of the elements. Nonconforming elements have a place in finite element analysis, but one must proceed carefully. Nonconforming elements are most useful for plates and shells, where continuity of first derivatives of displacement must be maintained for a conforming element, and that requirement leads to complicated formulations.

6.2 ELEMENT SHAPE LIMITS

Cubic elements in 3D and square elements or equilateral triangles in 2D generally result in equations that are well conditioned. If the element shape is greatly distorted from these ideal shapes, numerical difficulties can occur. Consequently, ANSYS has built-in checks on element shapes in the array of finite elements.[2] Such highly distorted elements may occur when the free mapping algorithm is used for meshing. They can be avoided by using mapped meshing

A warning will be generated if the element shape appears distorted. After a warning, you have the option to proceed with the analysis or to alter the element grid. An error message is generated and the calculation aborted if the element is so distorted that excessive numerical errors are likely. For 2D elements, ANSYS checks the aspect ratio, deviation from parallel sides for quadrilaterals, maximum corner angle, and the ratio of the Jacobian within quadrilaterals. Automatically generated meshes often result in warnings, and one has to decide whether to continue the analysis.

6.2.1 Aspect Ratio

For a triangle, Figure 6.1, the aspect ratio R for node n is defined by

$$R_n = \frac{2h_n}{a_n\sqrt{3}}, \tag{6.1}$$

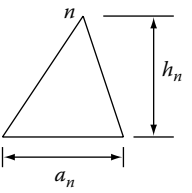


FIGURE 6.1 Aspect ratio for a triangle.

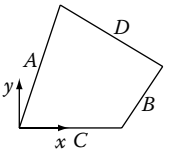


FIGURE 6.2 Quadrilateral element.

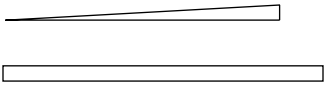


FIGURE 6.3 Badly distorted elements.

which has a value of 1 for an equilateral triangle. The aspect ratio for the element is the largest of the three possible ratios.

For the quadrilateral element (Figure 6.2), the aspect ratio is the ratio of the sides of a rectangle prescribed by the midpoints (A, B, C, D) of the sides:

$$r = \frac{X_B - X_A}{Y_D - Y_C}, \quad R = \max(r, r^{-1}). \tag{6.2}$$

Badly distorted elements are shown in Figure 6.3.

Warnings are issue for $R > 20$ although they may not preclude satisfactory results. An error is issued for $R > 10^6$ because round-off errors are likely. Round-off errors may occur for some computers even for $R = 10^3$.

6.2.2 Parallel Deviation for a Quadrilateral

The deviation from parallel opposite sides is measured by the angle between the sides (Figure 6.4). The maximum of the two angles is used as the measure of parallel deviation. Calculations may be degraded by highly distorted elements having a large parallel deviation (Figure 6.5).

ANSYS issues a warning if $\theta \geq 70°$ for elements without mid-side nodes and if $\theta \geq 100°$ for elements with mid-side nodes. Error statements are issued for angles greater than 150° or 170°, respectively.

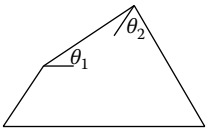


FIGURE 6.4 Measure of parallel deviation.

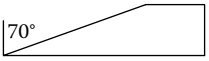


FIGURE 6.5 Parallel deviation of 70°.

6.2.3 Large Corner Angle

Another measure of element distortion is the enclosed angle at a corner (Figure 6.6).

ANSYS issues a warning if the maximum corner angle for a triangle exceeds 165°, or 155° for a quadrilateral without mid-side nodes, or 165° for a quadrilateral with mid-side nodes. An error is issued if the angle exceeds 179.9° in any case.

6.2.4 Jacobian Ratio

The determinant J of the Jacobian matrix 5.126 is a measure of the differential area in the isoparametric mapping for the quadrilateral,

$$dA = J d\xi d\eta, \tag{6.3}$$

and of the volume for the 3D element. For a square element, J is constant within the element. If J varies greatly within an element, the isoparametric mapping may become computationally unreliable. ANSYS calculates J at each element corner and at the centroid for 8-node quadrilaterals and 20-node brick elements. A warning is issued if the ratio of the maximum J to the minimum J for the element exceeds 30, and an error if the ratio exceeds 1000.

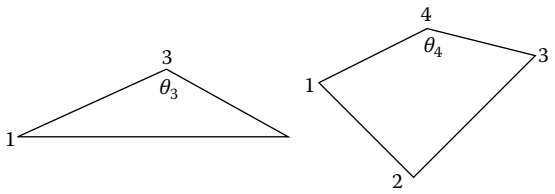


FIGURE 6.6 Corner angle for an element.

6.3 PATCH TEST

It is generally thought that convergence to the correct answer will occur if every patch of elements correctly models a constant strain condition. This is called the *patch test*, and should be applied to any element before using it.[3]

Consider any collection of contiguous elements, called a patch. Let $u_i^c, \varepsilon_{ij}^c, \tau_{ij}^c$, denote the fields of displacement, strain, and stress for a case of constant strain. We can calculate the nodal displacements $\mathbf{D}_K^c$ for the constant strain field from u_i^c by substituting the nodal coordinates.

The nodal forces for the finite element of an analysis can be calculated in two ways. First, using the stiffness matrix and the given nodal displacements $\mathbf{D}_K^c$, we find the corresponding nodal forces:

$$\mathbf{F} = \mathbf{K}\mathbf{D}^c. \tag{6.4}$$

Second, we can calculate the nodal forces that correspond to the boundary tractions T_i^c, which are in equilibrium with the stress field τ_{ij}^c. Using the tractions T_i^c, the applied nodal forces should be:

$$F_K^c = \sum_n \int_{S_{T_n}} T_i^c N_{iK} \mathrm{d}A. \tag{6.5}$$

If the patch of elements correctly models the state of constant strain, the two calculations of nodal force must yield the same result: $\mathbf{F} = \mathbf{F}^c$.

There are other equivalent forms of the patch test. One can specify the displacements only on the edge of the patch and calculate the nodal displacement at interior nodes, where $\mathbf{F} = 0$, and see if the interior nodes have displacements matching those generated by u_i^c. Alternatively, one could specify the nodal forces by $\mathbf{F} = \mathbf{F}^c$ and use the finite element equations to calculate $\mathbf{D}$, which is then compared to $\mathbf{D}^c$. This last procedure is often easier, since the finite element code is always programmed to accept loads as input. Naturally, one has to impose displacement boundary conditions that are sufficient to prevent rigid motion in order to solve the equations when the forces are given.

The patch test can be carried out by actually doing the calculations on the computer, or we can investigate the formulation of the problem to predict the result in the following way. The finite element equation is derived from the virtual work relation:

$$\sum \int_{V_n} \tau_{ij} \bar{\varepsilon}_{ij} \, \mathrm{d}V = \sum \int_{S_{T_n}} T_i^0 \bar{u}_i \, \mathrm{d}A + \sum \int_{V_n} b_i \bar{u}_i \, \mathrm{d}V, \tag{6.6}$$

for every set of virtual nodal displacements $\bar{D}_I$, with $\bar{\varepsilon}_{ij}$ and $\bar{u}_i$ generated by the shape functions:

$$\bar{u}_i = \sum_I N_{iI} \bar{D}_I, \quad \bar{e}_{ij} = \sum_I A_{ijI} \bar{D}_I. \tag{6.7}$$

For a constant strain field, $b_i = 0$ and $T_i = T_i^c$, so the right-hand side of 6.4 is equal to $\sum F_K^c \bar{D}_K$. The question now is whether the left-hand side gives that value for the chosen finite element model. The left-hand side of 6.6 can be integrated by parts over a region in which $\tau_{ij}\bar{u}_j$ is continuous by using the virtual work formula. We wish to include the nonconforming elements in this analysis and, for them, $\bar{u}_i$ is not continuous across element boundaries. Continuity will hold within each element so that, for each element, we have

$$\int_{\mathcal{V}'_m} \tau_{ij}\bar{\varepsilon}_{ij}\,\mathrm{d}V = \int_{\partial\mathcal{V}'_m} T_i\bar{u}_i\,\mathrm{d}A - \int_{\mathcal{V}'_m} \tau_{ij,j}\,\bar{u}_i\,\mathrm{d}V. \tag{6.8}$$

The integrals over the boundaries of the elements can be divided into the integral over the part that lies on the exterior boundary, if any, and the integral over the interior element boundary. Integrals over the common interior boundaries $\mathbf{S}_{nm}$ of the elements n and m can then be combined to obtain the following result

$$\sum_m \int_{\mathcal{V}_m} \tau_{ij}\bar{\varepsilon}_{ij}\,\mathrm{d}V = \sum \int_{S_{nm}} (T_i^n\,\bar{u}_i^n + T_i^m\,\bar{u}_i^m)\,\mathrm{d}A + \int_{S_T} T_i\bar{u}_i\,\mathrm{d}A - \int_{\mathcal{V}} \tau_{ij,j}\,\bar{u}_i\,\mathrm{d}V, \tag{6.9}$$

where we have used $\bar{u}_i = 0$ on any part S_u of the exterior boundary. Given $\tau_{ij} = \tau_{ij}^c$ and $T_i = T_i^c$, the last term is zero and the next to the last term is equal to $\sum F_K^c\delta_K$. Therefore,

$$\sum_m \int_{\mathcal{V}_m} \tau_{ij}^c e_{ij}\,\mathrm{d}V = \sum_K F_K^c\delta_K, \tag{6.10}$$

and the patch test is satisfied if the first term on the right-hand side is zero. That is,

$$\sum \int_{S_{nm}} (T_i^n\,\bar{u}_i^n + T_i^m\,\bar{u}_i^m)\,\mathrm{d}A = 0 \tag{6.11}$$

is a necessary condition for satisfying the patch test.

Let us next investigate this condition for conforming elements:

$$\bar{u}_i^n = \bar{u}_i^m = \bar{u}_i \quad \text{on} \quad S_{nm}. \tag{6.12}$$

In this case, the integrand of the term is zero if

$$T_i^n + T_i^m = 0 \quad \text{on} \quad S_{nm}. \tag{6.13}$$

Since the unit normals to the boundaries of adjacent elements are oppositely directed, this is the condition for equilibrium of tractions across element boundaries and is therefore satisfied when $\tau_{ij} = \tau_{ij}^c$. The conforming elements will therefore pass the patch test.

We see that 6.11 reduces to the condition for equilibrium of tractions at element boundaries if the displacements are continuous across the element boundaries. However, this reduction does not occur if the displacements are not continuous across the element boundary as is the case for nonconforming elements, and the term may not be zero for the constant strain state. We must explicitly evaluate this term.

If every element is given the stress and strain associated with a constant strain field, the differential equations of equilibrium and the stress boundary conditions will be satisfied, and the tractions T_i are constant for a given plane surface, having the same value for every element. Thus, if the element boundaries are plane, the integrals in 6.11 reduce to

$$\int_{\Sigma_{nm}} (T_i^n \bar{u}_i^n + T_i^m \bar{u}_i^m)\,\mathrm{d}A = T_i^n \int_{S_{nm}} (\bar{u}_i^n - \bar{u}_i^m)\,\mathrm{d}A. \tag{6.14}$$

Hence, the term will be zero if

$$\int_{S_{nm}} (\bar{u}_i^n - \bar{u}_i^m)\,\mathrm{d}A = 0 \tag{6.15}$$

for every interior element boundary. This integral has to be investigated for a nonconforming element.

Even if 6.15 is not exactly satisfied, the finite element solution may still converge to the correct solution if the integrals tend to zero with the size of the element. Of course, the terms must be zero for the actual loading, not just for constant strain. However, as the element size tends to zero, the actual state of strain can be approximated locally by a constant strain. Therefore, the test of a patch of elements in constant strain is thought to be a sufficient test for convergence.

6.3.1 Wilson–Taylor Quadrilateral

Let us consider, for example, the element discussed in Section 5.2.1—a rectangle with four nodeless parameters **d**.[4] If the nodeless parameters are nonzero for an isoparametric element, the displacements are not linear along an edge and continuity is lost. However, if **d** is zero in a particular case, then continuity is achieved. For each element, **d** is given by 5.46. Hence, **d** is zero if

$$\mathbf{k}_{21}\mathbf{D} = \iint \mathbf{B}^{\mathrm{T}}\mathbf{C}\mathbf{A}\mathbf{D}\,\mathrm{d}A = 0. \tag{6.16}$$

For the patch test, the nodal coordinates **D** are given the values such that **AD** is a matrix of constant strain. Since **CAD** is then a matrix of constant stress, it can be factored outside the integral sign and **d** will be zero if

$$\iint \mathbf{B}^{\mathrm{T}}\,\mathrm{d}A = \iint \mathbf{B}^{\mathrm{T}} J\,\mathrm{d}\xi\,\mathrm{d}\eta = 0. \tag{6.17}$$

For the rectangle, matrix **B** is given by 5.42 and contains only terms that are to the first power in ξ or η. Consequently, the integral 6.17 is indeed zero in this case. The displacement field is therefore continuous and the patch test will be satisfied.

If the shape functions are used to generate an isoparametric quadrilateral following the methods discussed in Section 5.5 with exact integration, the patch test fails unless the element is a parallelogram. However, it is possible to select the method of numerical integration so as to make $\mathbf{d} = 0$ for a constant strain field. For the general case, 5.37 has the form

$$u_\alpha = \sum_{k=1}^{4} N_{\alpha k} D_k + \sum_{a=1}^{4} N_{\alpha a}^{\text{ext}} d_a \tag{6.18}$$

where

$$\begin{aligned} N_{11}^{\text{ext}} &= N_{21}^{\text{ext}} = 1-\xi^2, \\ N_{12}^{\text{ext}} &= N_{22}^{\text{ext}} = 1-\eta^2, \end{aligned} \tag{6.19}$$

and

$$\varepsilon_{\alpha\beta} = \sum_{k=1}^{8} A_{\alpha\beta k} D_k + \sum_{a=1}^{4} B_{\alpha\beta a} d_a, \tag{6.20}$$

where

$$\begin{aligned} JB_{\alpha\beta a} &= \frac{J}{2}\left(\frac{\partial N_{\alpha a}^{\text{ext}}}{\partial x_\beta} + \frac{\partial N_{\beta a}^{\text{ext}}}{\partial x_\alpha} \right) \\ &= \frac{J}{2}\left(\frac{\partial N_{\alpha a}^{\text{ext}}}{\partial \xi_\theta} \frac{\partial \xi_\theta}{\partial x_\beta} + \frac{\partial N_{\beta a}^{\text{ext}}}{\partial \xi_\theta} \frac{\partial \xi_\theta}{\partial x_\alpha} \right), \\ &= \frac{1}{2}\left(\frac{\partial N_{\alpha a}^{\text{ext}}}{\partial \xi_\theta} JJ_{\theta\beta}^{*} + \frac{\partial N_{\beta a}^{\text{ext}}}{\partial \xi_\theta} JJ_{\theta\alpha}^{*} \right) \end{aligned} \tag{6.21}$$

$$\frac{\partial N_{11}^{\text{ext}}}{\partial \xi_1} = \frac{a}{2}\xi, \quad \frac{\partial N_{11}^{\text{ext}}}{\partial \xi_2} = 0, \text{ etc.} \tag{6.22}$$

The terms $\partial N_{\alpha a}^{\text{ext}} / \partial \xi_\theta$ are linear in (ξ,η), as are the terms $JJ_{\theta\beta}^{*}$. Their product is quadratic and therefore the integral of **B** over the area of the element may not be zero.

For the Wilson–Taylor element, the Jacobean terms $JJ^*_{\theta\beta}$ are evaluated at the centroid and are therefore constant. The components of $\boldsymbol{J}\mathbf{B}$ are then homogeneous linear functions of (ξ,η). The integral 6.17 will then integrate to zero and the patch test is satisfied. This modification is made in integration of all auxiliary matrices (5.44).

REFERENCES

1. Strang, G., and G. J. Fix, *An Analysis of the Finite Element Method*, Prentice-Hall, Englewood Cliffs, NJ, 1973.
2. *ANSYS Theory Reference Manual*, Section 13.7.
3. Bazeley, G. P., Y. K. Cheung, B. M. Irons, and O. C. Zienkiewicz, Triangular elements in plate bending–conforming and nonconforming solutions, *Proceedings of the First Conference on Matrix Methods in Structural Mechanics*, Wright-Patterson AFB, Ohio, 1965. See also Irons, B. M., E. A. de Oliveira, and O. C. Zienkiewicz, Comments on the paper: Theoretical foundations of the finite element method, by *Int. J. Solids Struct*., 6, 695–697, 1970.
4. Taylor, R. L., P. J. Beresford, and E. L. Wilson, A non-conforming element for stress analysis, *Int. J. Numer. Meth. Eng.*, 10, 1211–1219, 1976.

7 Heat Conduction in Elastic Solids

7.1 DIFFERENTIAL EQUATIONS AND VIRTUAL WORK

In this chapter, we will only consider small deformations and small temperature changes. Let T denote the change in temperature of a material particle from a reference temperature T_0. That is, the absolute temperature of the particle is $T + T_0$. For an elastic material with small temperature change, the stress–strain relation in matrix form is*

$$\begin{aligned} \boldsymbol{\tau} &= \mathbf{C}\boldsymbol{\varepsilon} - \boldsymbol{\beta}T, \\ \boldsymbol{\varepsilon} &= \mathbf{C}^{-1}\boldsymbol{\tau} + \boldsymbol{\alpha}T, \\ \boldsymbol{\beta} &= \mathbf{C}\boldsymbol{\alpha}. \end{aligned} \tag{7.1}$$

$\boldsymbol{\alpha}$ is the matrix of coefficients of linear expansion for the material. For an isotropic material $\boldsymbol{\alpha}^{\mathrm{T}} = \alpha[1\ 1\ 1\ 0\ 0\ 0]$ and $\boldsymbol{\beta} = \beta[1\ 1\ 1\ 0\ 0\ 0]$, $\beta = (2\mu + 3\lambda)\alpha$. The balance of energy provides the equation governing heart conduction at a material particle†

$$\rho c \frac{\partial T}{\partial t} = \nabla^{\mathrm{T}} \boldsymbol{\kappa} \nabla T + \rho r - T_0 \boldsymbol{\beta}^{\mathrm{T}} \frac{\partial \boldsymbol{\varepsilon}}{\partial t} \tag{7.2}$$

where c is the specific heat at constant strain, $\boldsymbol{\kappa}$ is the matrix of thermal conductivities, ρ is the mass density, and r is the energy supply at the element per unit mass. For an isotropic material, $\kappa_{ij} = \kappa\delta_{ij}$.

The gradient operator ∇ is a column matrix. In rectangular Cartesian coordinates,

$$\nabla^{\mathrm{T}} = \left[\frac{\partial}{\partial x} \quad \frac{\partial}{\partial y} \quad \frac{\partial}{\partial z} \right]. \tag{7.3}$$

In cylindrical coordinates,

$$\nabla^{\mathrm{T}} = \left[\frac{\partial}{\partial r} \quad \frac{1}{r}\frac{\partial}{\partial \theta} \quad \frac{\partial}{\partial z} \right]. \tag{7.4}$$

* Dill: Section 2.10. Refer to Preface.

† Dill: Equation 2.10.17. Refer to Preface.

The equivalent virtual work formula is generated by multiplying 7.2 by arbitrary functions $\bar{T}$ (i.e., a virtual temperature variation $\delta T = \bar{T}$), integrating over the volume, and applying integration by parts to obtain

$$\int_V \rho c \bar{T} \frac{\partial T}{\partial t} \mathrm{d}V + \int_V (\nabla \bar{T})^{\mathrm{T}} \boldsymbol{\kappa} \nabla T \, \mathrm{d}V + \int_V T_0 \bar{T} \boldsymbol{\beta}^{\mathrm{T}} \frac{\partial \boldsymbol{\varepsilon}}{\partial t} \mathrm{d}V = -\int_S \bar{T} q \, \mathrm{d}A + \int_V \bar{T} \rho r \, \mathrm{d}V, \tag{7.5}$$

where

$$q = -\mathbf{n}^{\mathrm{T}} \boldsymbol{\kappa} \nabla T \tag{7.6}$$

is the heat flux into the body across the surface with outward normal $\mathbf{n}$.

After dividing the volume into finite elements, we introduce shape functions for each element:

$$\mathbf{u} = \mathbf{N}\mathbf{D}, \ \boldsymbol{\varepsilon} = \mathbf{A}\mathbf{D}, \ T = \mathbf{S}\mathbf{T}, \ \nabla T = \mathbf{B}\mathbf{T}, \tag{7.7}$$

where $\mathbf{T}$ is the column matrix of nodal temperatures. Using the same shape functions to generate the virtual temperature variations yields the finite element form of the virtual work:

$$\mathbf{C}^{tt} \dot{\mathbf{T}} + \mathbf{K}^{tt} \mathbf{T} = \mathbf{Q} - \mathbf{C}^{tu} \dot{\mathbf{D}}, \tag{7.8}$$

where

$$\mathbf{K}^{tt} = \sum_{\text{elements}} \int_V \mathbf{B}^{\mathrm{T}} \boldsymbol{\kappa} \mathbf{B} \, \mathrm{d}V, \tag{7.9}$$

$$\mathbf{C}^{tt} = \sum_{\text{elements}} \int_V \rho c \, \mathbf{S}^{\mathrm{T}} \mathbf{S} \, \mathrm{d}V, \tag{7.10}$$

$$\mathbf{C}^{tu} = \sum_{\text{elements}} \int_V T_0 \mathbf{S}^{\mathrm{T}} \boldsymbol{\beta}^{\mathrm{T}} \mathbf{A} \, \mathrm{d}V, \tag{7.11}$$

$$\mathbf{Q} = -\sum_{\text{elements}} \int_S \mathbf{S}^{\mathrm{T}} q \, \mathrm{d}A + \sum_{\text{elements}} \int_V \mathbf{S}^{\mathrm{T}} \rho r \, \mathrm{d}V. \tag{7.12}$$

In the last equation, the surface integral includes only those segments of the surface that lie on the exterior boundary. Boundary conditions may provide nodal values of temperature $\mathbf{T}$, or the given surface heat transfer $q = q_i n_i$. If the heat transfer at the surface is by convective exchange with a surrounding fluid, then

$$q = -h(T_f - T), \tag{7.13}$$

where h is the film coefficient and T_f is the temperature of the fluid above the reference temperature of the element. The surface integral in 7.12 then includes the unknown nodal temperatures:

$$\sum_{\text{elements}} \int_S \mathbf{S}^{\mathrm{T}} q \, dA = \mathbf{K}^{tc}\mathbf{T} - \sum_{\text{elements}} \int_S h\mathbf{S}^{\mathrm{T}} T_f \, dA,$$
$$\mathbf{K}^{tc} = \sum_{\text{elements}} \int_S h\mathbf{S}^{\mathrm{T}}\mathbf{S}\,\mathrm{d}A. \tag{7.14}$$

The matrix $\mathbf{K}^{tc}$ can be added to the matrix $\mathbf{K}^{tt}$.

The complete heat conduction equations are coupled to the equations for the displacements by the last term in 7.8. The displacement equations are coupled to the temperature equations by the thermal stress $\boldsymbol{\tau} = -\boldsymbol{\beta}T$ in 7.1, which contributes internal nodal forces $\mathbf{F}^{th}$ according to 3.33. The mechanical equations including inertial and thermal forces are then

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{C}^{uu}\dot{\mathbf{D}} + \mathbf{K}^{uu}\mathbf{D} = \mathbf{F} - \mathbf{F}^{th}, \tag{7.15}$$

where

$$\mathbf{F}^{th} = \mathbf{K}^{ut}\mathbf{T},$$
$$\mathbf{K}^{ut} = -\sum_{\text{elements}} \int_V \mathbf{A}^{\mathrm{T}}\,\boldsymbol{\beta}\mathbf{S}\,\mathrm{d}V. \tag{7.16}$$

The coupled equations, 7.8 and 7.15, are

$$\begin{bmatrix} \mathbf{M} & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{D}} \\ \ddot{\mathbf{T}} \end{bmatrix} + \begin{bmatrix} \mathbf{C}^{uu} & 0 \\ \mathbf{C}^{tu} & \mathbf{C}^{tt} \end{bmatrix} \begin{bmatrix} \dot{\mathbf{D}} \\ \dot{\mathbf{T}} \end{bmatrix} + \begin{bmatrix} \mathbf{K}^{uu} & \mathbf{K}^{ut} \\ 0 & \mathbf{K}^{tt} \end{bmatrix} \begin{bmatrix} \mathbf{D} \\ \mathbf{T} \end{bmatrix} = \begin{bmatrix} \mathbf{F} \\ \mathbf{Q} \end{bmatrix}. \tag{7.17}$$

They must be solved simultaneously or by iteration. ANSYS provides elements SOLID5, PLANE13, AND SOLID98 for analysis of the transient thermoelastic coupled equations.

However, the coupling can often be neglected. Consider, for example, an isotropic material with constant κ so that the heat conduction equation (7.2) without radiation becomes

$$\rho c \frac{\partial T}{\partial t} = \kappa \nabla^2 T - T_0(2\mu + 3\lambda)\alpha \frac{\partial \varepsilon_{kk}}{\partial t}. \tag{7.18}$$

Suppose that the body expands freely ($\boldsymbol{\tau} = 0$), the stress–strain relation 7.1 gives $\varepsilon_{kk} = 3\alpha T$, and the heat conduction equation becomes

$$\begin{aligned} \left(1+f\right)\frac{\partial T}{\partial t} &= \frac{\kappa}{\rho c}\nabla^2 T, \\ f &= \frac{T_0\,(2\mu+3\lambda)\,3\,\alpha^2}{\rho c}. \end{aligned} \tag{7.19}$$

The nondimensional factor f is small for engineering materials. For a steel with

$$\rho = 8030 \text{ kg/m}^3, \qquad c = 503 \text{ J/kg } ^\circ\text{K},$$

$$\alpha = 17.82 \times 10^{-6}/^\circ\text{K}, \qquad \kappa = 16.3 \text{ J/s m } ^\circ\text{K},$$

$$E = 193 \text{ GPa}, \ \nu = 0.29, \quad T_0 = 293^\circ\text{K},$$

we find that $f = 0.03$. Consequently, the coupling of the equations through the strain in 7.2 can be neglected. The uncoupled equations can then be solved separately.

The temperature is first determined by the heat conduction equation:

$$\mathbf{C}^{tt}\dot{\mathbf{T}} + \mathbf{K}^{tt}\mathbf{T} = \mathbf{Q}. \tag{7.20}$$

The deformations are then subsequently determined by the thermoelastic equations:

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{C}^{uu}\dot{\mathbf{D}} + \mathbf{K}^{uu}\mathbf{D} = \mathbf{F} - \mathbf{F}^{th}. \tag{7.21}$$

ANSYS provides elements Plane55, Plane 77, Solid70, and Solid90 for the temperature calculations. The temperatures are saved for later stress analysis by

Load Step Options > Solution Printout,

which creates a file, jobname.rth. After the temperature analysis, the elements are switched to matching structural elements Plane 182, Plane 183, Solid185, or Solid186 for the thermal-stress analysis by the ETCHG command or

Element Type > Switch Element Type > Thermal to Structural.

The temperature is read into the displacement formulation by the LDREAD command or

Define Loads > Apply > Structural > Temperature > From Thermal Analysis.

7.2 EXAMPLE PROBLEM: ONE-DIMENSIONAL TRANSIENT HEAT FLUX

We will consider the uncoupled heat conduction problem with zero radiation supply for an isotropic material. From 7.18, the differential equation becomes

$$\rho c \frac{\partial T}{\partial t} = \kappa \nabla^2 T \tag{7.22}$$

We will use nondimensional variables:

$$\hat{x}_i = \frac{x_i}{a}, \quad \hat{T} = \frac{T}{T_0}, \quad \hat{t} = \frac{\kappa t}{\rho c a^2}. \tag{7.23}$$

Substitution into 7.22 and then dropping the ^, the basic equation of heat conduction becomes

$$\frac{\partial T}{\partial t} = \nabla^2 T \,. \tag{7.24}$$

Let us consider a problem similar to heating of a poker when you insert it into a fire in the fireplace. We will neglect the loss of heat on the sides and free end of the poker. We then have one-dimensional heat conduction:

$$\frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2}. \tag{7.25}$$

The initial temperature is taken to be zero on our nondimensional temperature scale, and the heated end has a temperature of 1 unit. In constructing a solution, it will be convenient to imagine a rod of double the length whose temperature is raised to that level on both ends. The middle of the body then has zero heat flux by symmetry. The double rod has a nondimensional length of 1: $0 \le x \le 1$. The initial conditions and the boundary conditions are therefore

$$\begin{aligned} &T(x,0) = 0, \\ &T(0,t) = 1, \\ &T(1,t) = 1. \end{aligned} \tag{7.26}$$

The steady-state solution is

$$T(x,\infty) = 1. \tag{7.27}$$

The transient part is

$$\theta(x,t) = T(x,t) - 1. \tag{7.28}$$

The equations to be solved are

$$\begin{aligned} &\frac{\partial \theta}{\partial t} = \frac{\partial^2 \theta}{\partial x^2}, \\ &\theta(x,0) = -1, \\ &\theta(0,t) = 0, \\ &\theta(1,t) = 0. \end{aligned} \tag{7.29}$$

The solution is easily determined by separation of variables to be

$$\theta(x,t) = -2\sum_{n\text{odd}} \mathrm{e}^{-n^2\pi^2 t} \sin(n\pi x). \tag{7.30}$$

The temperature at the free end of the original poker ($x = 0.5$) is

$$\begin{aligned} T\left(\frac{1}{2},t\right) &= 1 - \sum_{n\text{odd}} \mathrm{e}^{-n^2\pi^2 t} \frac{4}{n\pi} \sin\left(\frac{n\pi}{2}\right) \\ &= 1 - \sum_{n\text{odd}} \mathrm{e}^{-n^2\pi^2 t} \frac{4}{n\pi} (-1)^{(n-1)/2}. \end{aligned} \tag{7.31}$$

For the finite element analysis (see Section 15.30), the rod is represented as a plane rectangle of thickness 0.1 and unit length, meshed by 10 four-node thermal elements (QUAD 4 node 55). Nominal values of the nondimensional material properties are used: $\kappa = 1$, $\rho = 1$, and $c = 1$. The exponential function decays rapidly and the temperature reaches 99% of steady state after the nondimensional time is 0.5. A comparison of the exact solution from 7.31 with the numerical solution for $\Delta t = 0.01$ is shown in Tables 7.1 and 7.2. Note that the temperature is actually ramped up in the first numerical step so the theoretical instant step change in temperature is only approximated. One could take a very short first step then continue with larger steps.

TABLE 7.1
Temperature at $x = 0.5$

Time	Exact	FEA
0	0	0
0.05	0.228	0.227
0.1	0.526	0.505
0.2	0.823	0.805
0.3	0.934	0.923
0.4	0.975	0.970
0.5	0.991	0.988

TABLE 7.2
Temperature Distribution for $t = 0.1$

X	Exact	FEA
0	1	1
0.1	0.853	0.846
0.2	0.721	0.707
0.3	0.616	0.598
0.4	0.549	0.528
0.5	0.526	0.505

7.3 EXAMPLE: HOLLOW CYLINDER

A hollow cylinder is subjected to a temperature T_1 on the inner radius a and a temperature T_2 on the outer radius b (Figure 7.1). For a long cylinder, neglecting end effects, the temperature distribution is symmetric about the axis of the cylinder and depends only on the radius r. The steady-state distribution is found by solving

$$\nabla^2 T = 0 \tag{7.32}$$

using cylindrical coordinates:

$$T(r) = T_1 + (T_2 - T_1)\frac{\ln(r/a)}{\ln(b/a)}. \tag{7.33}$$

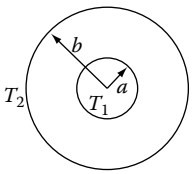


FIGURE 7.1 Heated cylinder.

The thermal stress problem (7.1) can then be solved for this temperature distribution. The displacement is radial and the stress distribution is symmetric about the axis of the cylinder. Assuming a plane strain condition for a long cylinder, one finds

$$u_r = \frac{\alpha(1+\nu)}{(1-\nu)r}\left(f(r) + \frac{(1-2\nu)r^2 + a^2}{b^2 - a^2} f(b) \right),$$

$$\tau_{rr} = \frac{\alpha E}{(1-\nu)r^2}\left(\frac{r^2 - a^2}{b^2 - a^2} f(b) - f(r) \right), \tag{7.34}$$

$$\tau_{\theta\theta} = \frac{\alpha E}{(1-\nu)r^2}\left(\frac{r^2 + a^2}{b^2 - a^2} f(b) + f(r) - r^2 T(r) \right),$$

where

$$f(r) = \int_a^r r\, T(r)\, \mathrm{d}r. \tag{7.35}$$

From

$$f(r) = T_1 \frac{r^2 - a^2}{2} + \frac{T_2 - T_1}{\ln(b/a)}\left(\frac{r^2}{2}\ln(r/a) - \frac{r^2 - a^2}{4} \right). \tag{7.36}$$

For a solution of this problem using ANSYS, see Section 15.32.

7.4 PROBLEMS

1. Use ANSYS to determine the temperature distribution for a square rod. The initial temperature is $T = 0$. The rod is then heated on both ends to a temperature $T = 1$. The nondimensional length is 1 and the width is 0.1. The nondimensional material properties are $\kappa = 1$, $\rho = 1$, and $c = 1$. The temperature rise at the middle is sought. Use the 4-node–55 thermal element. Treat the rod as a 2D body with no heat flux on the sides. Use a 1×10 grid. Submit a list of temperature at the middle of the rod at each time and a graph of T versus time. See Section 15.30.
2. Consider a solid cylinder with the axis along the z axis, $0 \le z \le L$. The cylinder is subjected to a given temperature T_1 on the end $z = 0$ and has zero heat flux q on the other surfaces. The proposed solution to the differential equation (7.22) has the form $T = f(z,t)$, where $f(0,t) = T_0$ and $\partial f(z,t)/\partial z = 0$ for $z = L$. Show that all boundary conditions are satisfied by this solution, that is, the temperature distribution is axisymmetric.

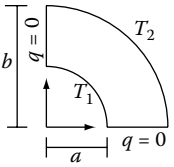


FIGURE 7.2 Heated disk.

3. An unrestrained cylindrical rod with initial temperature $T = 0$ is heated on one end to a temperature $T = 1$. The nondimensional radius is 0.1, the nondimensional length is 1, and the nondimensional material properties are $E = 1$, $\nu = 0.3$, $\alpha = 1$, $\kappa = 1$, $\rho = 1$, and $c = 1$. Heat loss on the exterior is neglected.
   (a) Determine the temperature distribution ($0 \leq t \leq 1$).
   (b) Determine the normal stress along the centerline at $t = 0.4$.

   Use the four-node thermal element Plane 55 for transient temperature calculation. The Plane 182 element is then automatically used for the thermal stress. See Section 15.31.
4. Solve the problem described in Section 7.3 using the finite element method. Determine the maximum value of $\tau_{\theta\theta}$ and the location, and compare the result with the exact solution from 7.34. Use actual dimensions and material properties: $a = 5$ in., $T_1 = 480°$, $T_2 = 75°$F, $E = 2.9 \times 10^7$ psi, $\nu = 0.3$, $\rho = 0.284$ lb/in.^{3}, $\alpha = 6.67 \times 10^{-6}$/°F, $\kappa = 8.092 \times 10^{-4}$ BTU/in.^{2} °F, $c = 0.1036$ BTU/lb°F. Use ¼ of the disk with zero circumferential heat flux (Figure 7.2). See Section 15.32.

8 Finite Element Method for Plasticity

8.1 THEORY OF PLASTICITY

The theory of plasticity describes the relation between load and deformation for many metals at higher stresses. The theory is rate independent: the same strains and stresses are obtained if the same loads are applied more rapidly (as long as inertial forces are negligible). The role of time in the constitutive equations is merely as a convenient bookkeeping parameter, and the occurrence of time derivatives does not indicate that such things as creep or relaxation can occur. The following is a summary of the basic equations.*

The material is initially elastic:

$$\tau(t)_{km} = c_{kmrs}\varepsilon(t)_{rs}, \varepsilon(t)_{km} = c^{-1}_{kmrs}\tau(t)_{rs}. \tag{8.1}$$

For an isotropic material,

$$c_{ijkm} = \mu(\delta_{ik}\delta_{jm} + \delta_{im}\delta_{jk}) + \lambda\delta_{ij}\delta_{km} \tag{8.2}$$

and

$$c^{-1}_{ijkm} = \frac{1+\nu}{2E}(\delta_{ik}\delta_{jm} + \delta_{im}\delta_{jk}) - \frac{\nu}{E}\delta_{ij}\delta_{km}. \tag{8.3}$$

When the stress—or equivalently, the strain—reaches a critical magnitude, the material yields and incremental deformations may be inelastic. If the magnitude of the stress then decreases, the incremental deformations are elastic. If the magnitude of the stress increases, the magnitude of the strain increases with apparently reduced moduli in a rate-independent manner. The additional strain,

$$\varepsilon^{\mathrm{P}}(t)_{ij} = \varepsilon(t)_{ij} - c^{-1}_{ijkm}\tau(t)_{km} \tag{8.4}$$

is called the plastic strain. Experiments on metals show that the volume change is elastic to a high degree of approximation, so that

$$\varepsilon^{\mathrm{P}}_{ii} = 0, \tag{8.5}$$

* Dill: Chapter 4. Refer to Preface.

and we will consider only this case. Typical criteria for yielding have the form

$$f(\boldsymbol{\tau}(t),\boldsymbol{\varepsilon}^{\mathrm{P}}(t),\boldsymbol{\beta}(t),\kappa(t)) \leq 0 \tag{8.6}$$

where the tensor $\boldsymbol{\beta}$ is called the back stress and the scalar κ is called the hardening parameter. Both parameters depend on the history of plastic strain. The yield function f is normalized so that it is negative before yield, and yield occurs when $f = 0$. For fixed $\boldsymbol{\varepsilon}^{\mathrm{P}}$, $\boldsymbol{\beta}$, and κ, the condition $f = 0$ defines a surface in stress space, which is called the yield surface. Experimental evidence shows that this surface is closed and convex, and we will also assume that it is smooth. The tensor

$$\eta_{ij} = \frac{\partial f}{\partial \tau_{ij}} \tag{8.7}$$

is an outward normal to the yield surface. Suppose the material is at the point of yield so that $f(t) = 0$. An increment of stress inward from the yield surface reduces the magnitude of f and so an elastic increment in strain occurs. An increment in stress that is outward from the surface expands the yield surface and plastic strains occur. Let us define

$$\hat{f} = \eta_{ij}\dot{\tau}_{ij}. \tag{8.8}$$

At yield, $f = 0$, we have three possibilities:

(1) Elastic unloading: $\hat{f} < 0$ and therefore $\dot{\boldsymbol{\varepsilon}}^{\mathrm{P}} = 0$.
(2) Plastic loading: $\hat{f} > 0$ and therefore $\dot{\boldsymbol{\varepsilon}}^{\mathrm{P}} \neq 0$.
(3) A neutral process: $\hat{f} = 0$ and $\dot{\boldsymbol{\varepsilon}}^{\mathrm{P}} = 0$.

For elastic unloading and the neutral process the incremental strain is elastic.

Typical constitutive relations for the back stress and the hardening have the form

$$\begin{aligned} \dot{\boldsymbol{\beta}}(t) &= \mathbf{b}(\boldsymbol{\tau}(t), \boldsymbol{\varepsilon}^{\mathrm{P}}(t), \boldsymbol{\beta}(t), \kappa(t))\hat{f}(t), \\ \dot{\kappa}(t) &= k(\boldsymbol{\tau}(t), \boldsymbol{\varepsilon}^{\mathrm{P}}(t), \boldsymbol{\beta}(t), \kappa(t))\hat{f}(t). \end{aligned} \tag{8.9}$$

The plastic strain increment during plastic loading is determined by

$$\dot{\varepsilon}_{ij}^{\mathrm{P}} = \eta_{ij}\frac{\hat{f}}{2\bar{G}} \tag{8.10}$$

where $\bar{G}$ is a plastic modulus that depends on the history of plastic straining. Using 8.4, we find for plastic loading that

$$\dot{\varepsilon}(t)_{ij} = \left(c_{ijkm}^{-1} + \frac{1}{2\bar{G}}\eta_{ij}\eta_{km}\right)\dot{\tau}(t)_{km}\,. \tag{8.11}$$

This equation can be inverted to obtain

$$\dot{\tau}_{ij} = \bar{c}_{ijkm}\dot{\varepsilon}_{km} \tag{8.12}$$

where the tangent modulus tensor is

$$\begin{aligned} \bar{c}_{ijkm} &= c_{ijkm} - \frac{1}{2\mu g} c_{ijrs}\eta_{rs}\eta_{pq}c_{pqkm}, \\ 2\mu g &= 2\bar{G} + c_{ijrs}\eta_{ij}\eta_{rs}. \end{aligned} \tag{8.13}$$

For an isotropic material,

$$\begin{aligned} \bar{c}_{ijkm} &= c_{ijkm} - \frac{2\mu}{g}\eta_{ij}\eta_{km}, \\ 2\mu g &= 2\bar{G} + 2\mu\eta_{ij}\eta_{ij}. \end{aligned} \tag{8.14}$$

A commonly used set of specific constitutive relations is the following

$$\begin{aligned} & f(\boldsymbol{\tau}(t), \boldsymbol{\varepsilon}^{\mathrm{P}}(t), \boldsymbol{\beta}(t), \kappa(t)) = \bar{\sigma}(t) - \kappa(t), \\ & \bar{\sigma} = +\sqrt{\frac{3}{2}s_{ij}s_{ij}}, \\ & \boldsymbol{\beta} = 0, \\ & \kappa = H(\bar{\varepsilon}^{\mathrm{P}}), \\ & \dot{\bar{\varepsilon}}^{\mathrm{P}} = +\sqrt{\frac{2}{3}\dot{\varepsilon}_{ij}^{\mathrm{P}}\dot{\varepsilon}_{ij}^{\mathrm{P}}}. \end{aligned} \tag{8.15}$$

This is called the von Mises yield criterion with isotropic strain hardening, where $\bar{\sigma}$ is the effective stress, which is also called the von Mises stress, and $\bar{\varepsilon}^{\mathrm{P}}$ is the effective plastic strain. The tensor $\mathbf{s}$ is the deviatoric stress:

$$s_{ij} = \tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij}. \tag{8.16}$$

In this case,

$$\eta_{ij} = \frac{3}{2}\frac{s_{ij}}{\bar{\sigma}}, \text{ and } \eta_{ij}\eta_{ij} = \frac{3}{2}. \tag{8.17}$$

From 8.8,

$$\hat{f} = \frac{3}{2}\frac{s_{ij}\dot{\tau}_{ij}}{\bar{\sigma}}. \tag{8.18}$$

Using 8.10,

$$\dot{\bar{\varepsilon}}^{\mathrm{P}} = \frac{\hat{f}}{2\bar{G}} . \tag{8.19}$$

At yield,

$$\bar{\sigma} = H(\bar{\varepsilon}^{\mathrm{P}}) . \tag{8.20}$$

Using 8.10, the condition $\dot{f} = 0$ for plastic loading gives

$$2\bar{G} = H'. \tag{8.21}$$

From 8.10, for plastic loading

$$\dot{\varepsilon}_{ij}^{\mathrm{P}} = \frac{9 s_{ij} s_{km} \dot{\tau}_{km}}{4H'H^2} . \tag{8.22}$$

For an isotropic material,

$$2\mu g = H' + 3\mu \tag{8.23}$$

and

$$\bar{c}_{ijrs} = c_{ijrs} - \frac{9\mu^2 s_{ij} s_{rs}}{(H' + 3\mu)H^2} . \tag{8.24}$$

The material is completely characterized by the elastic constants and the one function H. These can be determined by the tensile test.

8.1.1 Tensile Test

In this test, $\tau_{11} > 0$ and the other components of stress are zero. Therefore, $\bar{\sigma} = \tau_{11}$. Since the plastic volume change is zero, $\varepsilon_{22}^{\mathrm{P}} = \varepsilon_{33}^{\mathrm{P}} = -\frac{1}{2}\varepsilon_{11}^{\mathrm{P}}$ and $\bar{\varepsilon}^{\mathrm{P}} = \varepsilon_{11}^{\mathrm{P}}$, for loading from the virgin state. The yield criterion then becomes $\tau_{11} = H(\varepsilon_{11}^{\mathrm{P}})$. This function can be measured as illustrated in Figure 8.1.

For the finite element analysis, one must describe the function H numerically. This is usually done via a piecewise linear function. The simplest approximation is by one straight line (Figure 8.2). This is called the bilinear model of plasticity. If E is the elastic modulus, the slope of the hardening portion of the stress–strain curve is some fraction m of E as shown in Figure 8.2. Then,

$$H(\varepsilon^{\mathrm{P}}) = kE\varepsilon^{\mathrm{P}} + Y, \quad k = \frac{m}{1-m}. \tag{8.25}$$

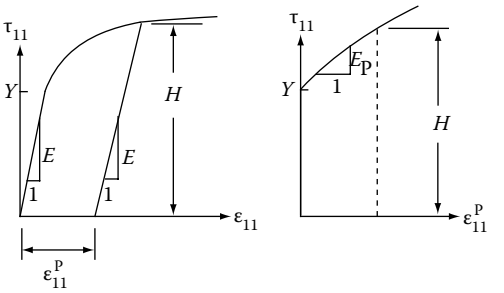


FIGURE 8.1 Tensile test.

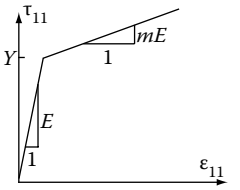


FIGURE 8.2 Bilinear model.

The case $m = 0$ is called perfectly plastic. This model is not very accurate but it does provide the essential qualitative features of plastic behavior.

8.1.2 Plane Stress

For practical applications, we often need the specialization of the theory to the plane problem. The equations are directly applicable to plane strain by setting $\varepsilon_{k3} = 0$. The case of plane stress is more complicated, and will be treated now. For an isotropic material, 8.11 becomes

$$\dot{\varepsilon}_{ij} = \frac{1}{2\mu}\dot{\tau}_{ij} - \frac{\nu}{(1+\nu)}\frac{\dot{\tau}_{kk}}{2\mu}\delta_{ij} + \frac{9 s_{ij} s_{km}}{4H'H^2}\dot{\tau}_{km}. \tag{8.26}$$

The condition for plane stress is $\tau_{k3} = 0$. Thus,

$$\dot{\varepsilon}_{\alpha\beta} = \frac{1}{2\mu}\dot{\tau}_{\alpha\beta} - \frac{\nu}{(1+\nu)}\frac{\dot{\tau}_{\phi\phi}}{2\mu}\delta_{\alpha\beta} + \frac{9 s_{\alpha\beta} s_{\theta\phi}}{4H'H^2}\dot{\tau}_{\theta\phi}, \tag{8.27}$$

where the Greek indices range over (1, 2). The inverse of this relation can be expressed as follows.

$$\frac{1}{2\mu}\dot{\tau}_{\alpha\beta} = \dot{\varepsilon}_{\alpha\beta} + \frac{\nu}{1-\nu}\dot{\varepsilon}_{\theta\theta}\delta_{\alpha\beta} - \frac{1}{\eta}f_{\alpha\beta}f_{\theta\phi}\dot{\varepsilon}_{\theta\phi},$$

$$f_{\alpha\beta} = s_{\alpha\beta} + \frac{\nu}{1-\nu}s_{\theta\theta}\delta_{\alpha\beta}, \tag{8.28}$$

$$\eta = \frac{2H'H^2}{9\mu} + f_{\alpha\beta}s_{\alpha\beta}.$$

This relation holds only for plane stress.

8.1.3 Summary of Plasticity

The solution procedure is as follows. Having determined the state of stress and the plastic strain up to time t, we determine the current incremental stress–strain relation as follows. First, use the yield criterion to test whether the material at each point has yielded. If not, use the elastic stress–strain law. If the material at a point has reached yield, there are two possibilities depending on whether the future will be loading or unloading. If the material has yielded but $s_{km}\dot{\tau}_{km} \le 0$, no plastic flow occurs and the elastic stress–strain law is still used. If the material has yielded and $s_{km}\dot{\tau}_{km} > 0$, plastic flow occurs and one must use equation. In matrix form,

$$\dot{\boldsymbol{\tau}} = \mathbf{C}_{\mathrm{T}}\dot{\boldsymbol{\varepsilon}}, \tag{8.29}$$

The coefficient matrix $\mathbf{C}_{\mathrm{T}}$ is aptly called the tangent modulus matrix. The elements of $\mathbf{C}_{\mathrm{T}}$ are given by

$$\mathbf{C}_{\mathrm{T}} = \mathbf{C} - \mathbf{C}_{\mathrm{P}}, \tag{8.30}$$

where $\mathbf{C}$ is the usual matrix of elastic coefficients and $\mathbf{C}_{\mathrm{P}}$ is a reduction due to plastic flow.

In the case of isotropic strain hardening and the von Mises yield criterion, a tensile test on the material determines the elastic constants E and ν (and therefore, λ and μ), the initial yield stress Y, and the strain hardening function $H(\bar{\varepsilon}^{\mathrm{P}})$. If the bilinear model fits the data satisfactorily, the representation 8.25 can be used.

For general three-dimensional problems, 8.24 gives immediately

$$\mathbf{C}_{\mathrm{P}} = \frac{9\mu^2}{(H'+3\mu)\tau_E^2}\begin{bmatrix} s_{11}s_{11} & s_{11}s_{22} & s_{11}s_{33} & s_{11}s_{12} & s_{11}s_{23} & s_{11}s_{31} \\ & s_{22}s_{22} & s_{22}s_{33} & s_{22}s_{12} & s_{22}s_{23} & s_{22}s_{31} \\ & & s_{33}s_{33} & s_{33}s_{12} & s_{33}s_{23} & s_{33}s_{31} \\ & & & s_{12}s_{12} & s_{12}s_{23} & s_{12}s_{31} \\ & \text{sym} & & & s_{23}s_{23} & s_{23}s_{31} \\ & & & & & s_{31}s_{31} \end{bmatrix}. \tag{8.31}$$

In this case of plane strain, this formula reduces to

$$\mathbf{C}_{\mathrm{P}} = \frac{9\mu^2}{(H' + 3\mu)\tau_E^2} \begin{bmatrix} s_{11}s_{11} & s_{11}s_{22} & s_{11}s_{12} \\ s_{22}s_{11} & s_{22}s_{22} & s_{22}s_{12} \\ s_{12}s_{11} & s_{12}s_{22} & s_{12}s_{12} \end{bmatrix}. \tag{8.32}$$

For plane stress, from (8.1.28),

$$\mathbf{C}_{\mathrm{P}} = \frac{2\mu}{\eta} \begin{bmatrix} f_{11}f_{11} & f_{11}f_{22} & f_{11}f_{12} \\ f_{22}f_{11} & f_{22}f_{22} & f_{22}f_{12} \\ f_{12}f_{11} & f_{12}f_{22} & f_{12}f_{12} \end{bmatrix}. \tag{8.33}$$

The plastic modulus $\mathbf{C}_{\mathrm{P}} = 0$ if the stress level is below yield. Once yielding occurs, $\mathbf{C}_{\mathrm{P}}$ depends on the history of plastic strain. The constitutive relation determines only the rate of change in stress, so one must integrate over the history to obtain the total stress. In practice, we only know the direction of the external loads at time t. We may not know whether loading or unloading will occur for every point of the body. It is often necessary to proceed by trial and error: make an assumption at each point of loading or unloading, do the calculations of the incremental stress, and check to see if the assumption was correct.

8.2 FINITE ELEMENT FORMULATION FOR PLASTICITY

The fundamental problem is to find the displacement field such that the stress determined by the constitutive relations will satisfy the equations of equilibrium and the load conditions at all times. We will proceed step by step to trace the history of deformation for a given sequence of loads. We will use matrix notation.

The displacement field within each element is described with the aid of shape functions in the same way as for elasticity:

$$\mathbf{u} = \mathbf{ND}. \tag{8.34}$$

The strain field is still determined by the gradient of displacement so that

$$\boldsymbol{\varepsilon} = \mathbf{AD}. \tag{8.35}$$

The shape functions $\mathbf{N}$ and the geometric matrix $\mathbf{A}$ are characteristics of the finite element model and are independent of the history of deformation.

The theorem of minimum potential energy does not apply in the theory of plasticity because of the path-dependent nature of the constitutive relations. However, the theorem of virtual work does apply since that theorem is independent of the constitutive assumption. The governing equations for the rate of displacement are

easily found by direct use of the virtual work formula. In matrix notation, with $\mathcal{S}$ the boundary of $\mathcal{V}$,

$$\int_{\mathcal{V}} \bar{\boldsymbol{\varepsilon}}^{\mathrm{T}} \boldsymbol{\tau} \, dV = \int_{\mathcal{S}} \bar{\mathbf{u}}^{\mathrm{T}} \mathbf{T} \, dA + \int_{\mathcal{V}} \bar{\mathbf{u}}^{\mathrm{T}} \mathbf{b} \, dV. \tag{8.36}$$

This relation must hold for every compatible system of displacements $\bar{\mathbf{u}}$ and strains $\bar{\boldsymbol{\varepsilon}}$. Compatible systems can be generated by using relations 8.34 and 8.35 where the nodal displacements are given arbitrary values: $\bar{\mathbf{u}} = \mathbf{N}\bar{\mathbf{D}}$ and $\bar{\boldsymbol{\varepsilon}} = \mathbf{A}\bar{\mathbf{D}}$. Substituting these relations in 8.36 and requiring that the equation holds for arbitrary nodal displacements $\bar{\mathbf{D}}$ gives the following simple balance of forces for the finite element model:

$$\mathbf{f} = \mathbf{F}, \tag{8.37}$$

where

$$\mathbf{F} = \sum_{n} \int_{\mathcal{S}_n} \mathbf{N}^{\mathrm{T}} \mathbf{T} \, dA + \sum_{n} \int_{\mathcal{V}_n} \mathbf{N}^{\mathrm{T}} \mathbf{b} \, dV, \tag{8.38}$$

and

$$\mathbf{f} = \sum_{n} \int_{\mathcal{V}_n} \mathbf{A}^{\mathrm{T}} \boldsymbol{\tau} \, dV. \tag{8.39}$$

The force matrix $\mathbf{F}$ is the matrix of nodal loads (and reactions) at any instant calculated from the surface tractions and body force in the usual manner: The surface integral extends over that part of the element that forms the exterior boundary, if any. Matrix $\mathbf{f}$ is the matrix of resisting nodal forces due to the internal stress state. Equation 8.37 merely states that the internal nodal forces balance the external nodal forces when the stresses are in equilibrium. The only time-dependent quantities in the formulas for $\mathbf{f}$ and $\mathbf{F}$ are the stress $\boldsymbol{\tau}$, the tractions $\mathbf{T}$ and possibly the body force $\mathbf{b}$.

8.2.1 Fundamental Solution

The balance of forces (8.37) holds at every instant, but a direct solution is not possible because we do not have a constitutive relation for the total stress. The constitutive relation for plasticity only determines the rate of stress:

$$\dot{\boldsymbol{\tau}} = \mathbf{C}_{\mathrm{T}} \dot{\boldsymbol{\varepsilon}}. \tag{8.40}$$

It is necessary to proceed step by step in time to trace the history of deformation. This is unavoidable whenever we have inelastic or nonlinear material behavior.

If the displacements and stresses are known at time t_n, the solution can be marched forward by the approximation

$$\mathbf{D}(t_{n+1}) = \mathbf{D}(t_n) + \dot{\mathbf{D}}(t_n)\mathrm{d}t \tag{8.41}$$

$$\boldsymbol{\tau}(t_{n+1}) = \boldsymbol{\tau}(t_n) + \dot{\boldsymbol{\tau}}(t_n)\mathrm{d}t, \tag{8.42}$$

where $\mathrm{d}t = t_{n+1} - t_n$. Therefore, we proceed to develop a solution procedure for determining the increments. The basic relation governing the increments is just the time derivative of the balance of forces (8.37). The value of $\dot{\mathbf{F}}$ is the given time rate of loading. The value of $\dot{\mathbf{f}}$ is found via the time derivative of 8.39 with $\dot{\boldsymbol{\tau}}$ given by 8.40 and $\dot{\boldsymbol{\varepsilon}}$ determined via the time derivative of 8.35 for each element. The result is $\dot{\mathbf{f}} = \mathbf{K}_{\mathrm{T}}\dot{\mathbf{D}}$. Thus,

$$\mathbf{K}_{\mathrm{T}}\dot{\mathbf{D}} = \dot{\mathbf{F}}, \tag{8.43}$$

where the stiffness matrix is the merge of the element matrices given by

$$\mathbf{K}_{\mathrm{T}} = \int_{\mathcal{V}} \mathbf{A}^{\mathrm{T}}\mathbf{C}_{\mathrm{T}}\mathbf{A}\mathrm{d}V. \tag{8.44}$$

These element stiffness matrices depend on the history of strain through the tangent modulus $\mathbf{C}_{\mathrm{T}}$. In the case of plane stress or plane strain, the volume integral reduces to the integral over the area.

This step-by-step solution can be made as accurate as desired by decreasing the step size (up to the limit imposed by the finite element idealization and the numerical errors). However, each step requires that the stiffness matrix be reformed and the equilibrium equations 8.43 must be solved.

In this fundamental procedure, $\mathbf{C}_{\mathrm{P}}$ is corrected only at the end of each step, and the elements that yield in that step will have been assigned a stiffness that is too large (or too small for unloading). The error accumulates with each step and the final solution will have a significant error unless the error in each time step is very small. One either has to use a very large number of time steps (very small increment in load) or else the solution for each step has to be improved over the accuracy obtained with the simple formula, 8.41.

8.2.2 Iteration to Improve the Solution

The accuracy of each step can be improved, and therefore the need for small steps reduced, by iteration within each step. The general idea is an extension of the Newton–Raphson method of solution of a nonlinear algebraic equation. For any increments in stress and strain, to first order, we have

$$\mathrm{d}\boldsymbol{\tau} = \mathbf{C}_{\mathrm{T}}\mathrm{d}\boldsymbol{\varepsilon}. \tag{8.45}$$

Suppose that some approximation $\mathbf{D}^i$ to the solution up to a particular time t has been found. We now wish to improve the solution by finding improved displacements $\mathbf{D}^{i+1} = \mathbf{D}^i + \mathrm{d}\mathbf{D}^i$. For the increment $\mathrm{d}\mathbf{D}^i$ in nodal displacements, the strains in an element will be changed according to formula 8.35,

$$\mathrm{d}\varepsilon^i = \mathbf{A}\mathrm{d}\mathbf{D}^i, \tag{8.46}$$

and the stress will be changed by 8.45. Thus,

$$\mathrm{d}\boldsymbol{\tau}^i = \mathbf{C}_{\mathrm{T}}^i \mathrm{d}\boldsymbol{\varepsilon}^i = \mathbf{C}_{\mathrm{T}}^i \mathbf{A}\mathrm{d}\mathbf{D}^i, \tag{8.47}$$

where the tangent modulus is evaluated for the stress $\boldsymbol{\tau}^i$ corresponding to the deformation history up to $\mathbf{D}^i$. The internal node forces for the ith iteration are given by 8.39 using the stress $\boldsymbol{\tau}^i$. The change in internal node forces for the i and $i + 1$ approximations is therefore

$$\mathrm{d}\mathbf{f}^i = \mathbf{f}^{i+1} - \mathbf{f}^i = \sum \int_{V_n} \mathbf{A}^{\mathrm{T}}\, \mathrm{d}\boldsymbol{\tau}^i\, \mathrm{d}V. \tag{8.48}$$

After calculating the internal nodal forces $\mathbf{f}^i$ for the ith iteration, we seek a correction such that the internal node forces $\mathbf{f}^{i+1}$ are equal to the applied loads $\mathbf{F}$. Using 8.47, we see that the right-hand side of 8.48 is the tangent stiffness matrix times the incremental displacements. Thus,

$$\mathbf{K}_{\mathrm{T}}^i \mathrm{d}\mathbf{D}^i = \mathbf{F} - \mathbf{f}^i. \tag{8.49}$$

The superscript i means that the term is evaluated for the ith iteration. The right-hand side is the unbalanced node force. $\mathbf{F}$ is the total applied load at the instant for which the iteration is taking place. The internal resisting node forces $\mathbf{f}^i$ are given by

$$\mathbf{f}^i = \sum \int_{\mathbf{V}_n} \mathbf{A}^{\mathrm{T}}\, \boldsymbol{\tau}^i\, \mathrm{d}V. \tag{8.50}$$

The process is started by using the fundamental step-by-step process to generate a first approximation $\mathbf{D}^1$ for the displacements at time $t + \mathrm{d}t$. Equation 8.49 is then solved for $\mathrm{d}\mathbf{D}^1$ and so forth. The displacement

$$\mathbf{D}^{i+1} = \mathbf{D}^i + \mathrm{d}\mathbf{D}^i \tag{8.51}$$

is an improved approximation.

Equation 8.49 describes the fundamental procedure. The stiffness matrix is reformed and the equilibrium equations are then solved at each iteration. One may, however, approximate $\mathbf{K}_{\mathrm{T}}^{i}$ by the tangent stiffness matrix for some particular time, say $\mathbf{K}_{\mathrm{T}}^{i} = \mathbf{K}_{\mathrm{T}}(t)$, and keep it fixed throughout the iteration, or it may be corrected (updated) only after some fixed number of iterations. The process is a form of the modified Newton–Raphson method.

8.3 EXAMPLE: SHORT BEAM

The plane stress problem examined in Section 2.1.14 and shown in Figure 8.3, will be solved for the case of the bilinear material model with the following material properties.

$$E = 30 \times 10^3 \text{ ksi}, \nu = 1/3, Y = 30 \text{ ksi}, k = 0.1$$

The four-node rectangle is used with 50 segments per side. That gives 2500 elements and 2601 nodes. In the actual analysis, the symmetry about the mid-plane $x = 0.5$ is used so that the actual number of elements is reduced by one-half (see Section 15.13).

In this problem, the singularity at the built-in corner means that the yield limit will be immediately exceeded at the corner for any nonzero load. However, the plastic zone is very localized for small loads. The finite element analysis for any finite number of elements gives a finite approximate solution for the maximum stress in the corner, and the calculated behavior is elastic until that stress reaches the yield point.

After an element reaches the yield stress, one must proceed step by step, or by iteration, for the subsequent loading. Up to five iterations were used in each subsequent step, as needed to achieve the set error level. The calculated corner displacement is shown in Figure 8.4 for a monotonically increasing load.

As the load exceeds 5 ksi, increasing softening due to plastic flow can be seen. The corner element reaches the yield stress first, then the adjacent elements yield, and as the load increases the region of yielding grows outward from the corner. The

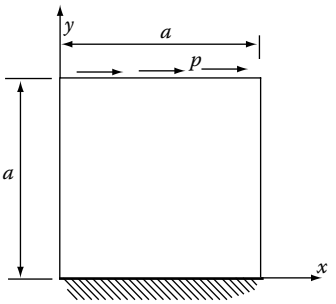


FIGURE 8.3 Plane stress: short beam.

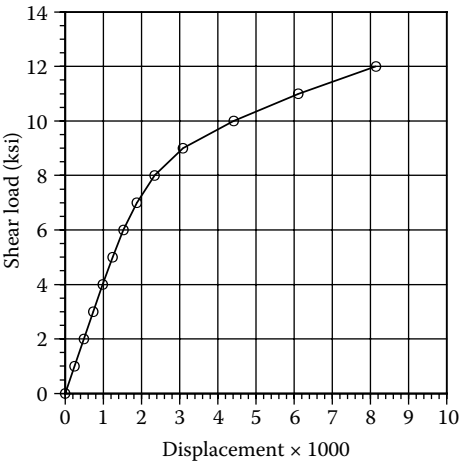


FIGURE 8.4 Plane stress example: short beam.

stresses at the integration points nearest the fixed boundary are shown in Figures 8.5 to 8.7 for the initial yield and two subsequent higher loads. The shoulder in the curve for σ_x occurs at the boundary of the yielded region. Since this material has nonzero strain hardening, the stress in the yielded region increases toward the boundary. However, the stresses in the yielded region grow more slowly because of the softening of the yielded material. Very near the corner, the stress increases rapidly as it does near the singularity in the elasticity problem.

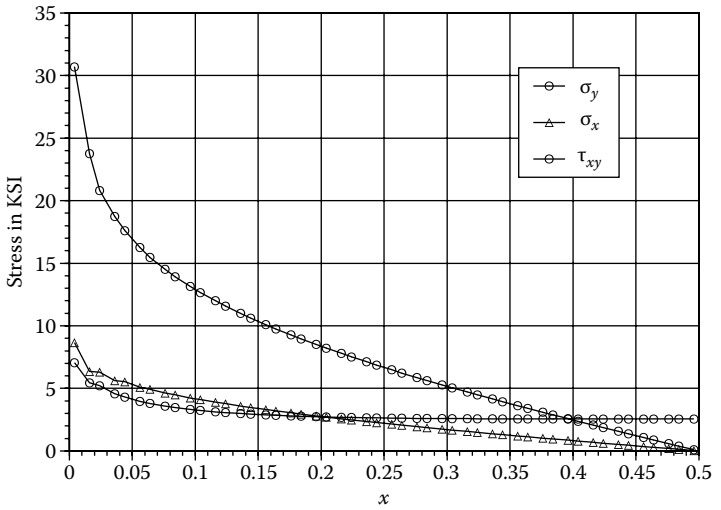


FIGURE 8.5 Short beam: stress at $y = 0.004$ for a load of 2.9 ksi.

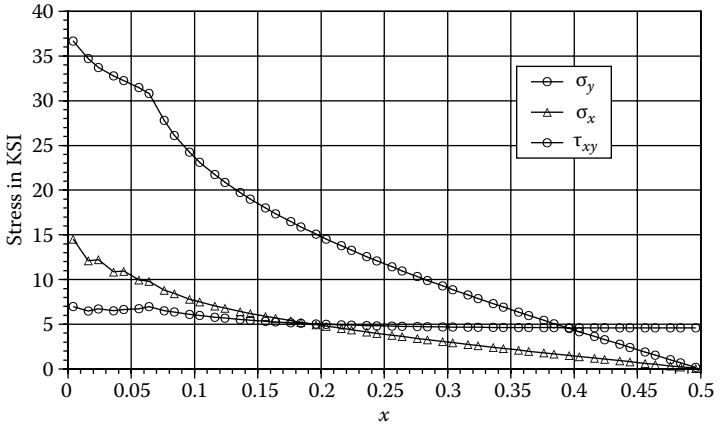


FIGURE 8.6 Short beam: stress at $y = 0.004$ for a load of 5 ksi.

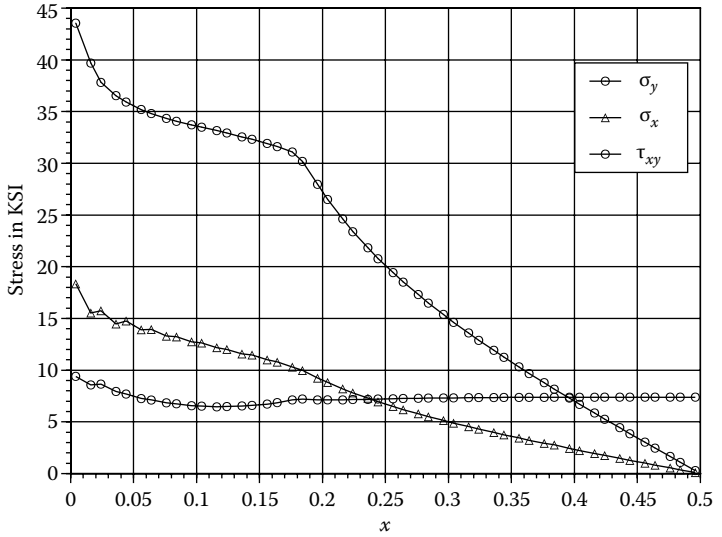


FIGURE 8.7 Short beam: stress at $y = 0.004$ for a load of 7 ksi.

8.4 PROBLEMS

1. In a tensile test, $\tau_{11} = \sigma$ and the other stress components are zero. Show this for the case of loading of a material with a von Mises yield criterion and isotropic strain hardening.
2. For the bilinear plasticity model (Figure 8.8), show that the hardening law is

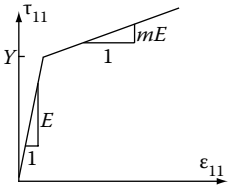


FIGURE 8.8 Bilinear model.

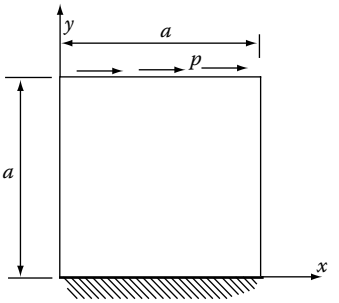


FIGURE 8.9 Short beam.

$$H(\varepsilon^{\mathrm{P}}) = kE\varepsilon^{\mathrm{P}} + Y, \quad k = \frac{m}{1-m}.$$

3. Solve the plane stress problem shown in Figure 8.9 for an elastic–plastic material with von Mises yield and isotropic hardening with a bilinear model $a = 1$, $p = 10{,}000$, $E = 30 \times 10^6$, $\nu = 1/3$, $Y = 30{,}000$, $E_{\mathrm{T}} = 2.727 \times 10^6$. Use a 20 by 20 grid of 400 elements (see Section 15.13). Apply the load in 10 steps ($0 \le t \le 1$). Then remove the load ($1 \le t \le 2$). Submit:
   (a) A list of UX at the corner for each time step $0 \le t \le 2$.
   (b) A plot of the normal stress along the base at maximum load ($t = 1$).
   (c) A plot of the residual normal stress along the base after unloading ($t = 2$).

Note: Be sure to adjust the colors to get a white background on your graphs.

4. Consider small deformations of a nonlinearly elastic material: $\boldsymbol{\tau} = f(\varepsilon)$. Derive the finite element equations for solution of the small displacement equations by the iterative procedure for a general element and shape functions.
5. Consider small deformations of a material such that $\tau_{ij} = a_{ijkm}\varepsilon_{km} + b_{ijkmrs}\varepsilon_{km}\varepsilon_{rs}$. Derive a solution procedure by the finite element method for a general element and shape functions.

BIBLIOGRAPHY

Khan, A. S., and Huang, S., *Continuum Theory of Plasticity*, John Wiley & Sons, New York, 1995.

Lubliner, J., *Plasticity Theory*, Macmillan Publ. Co., New York, 1990.

Lubarda, V. A., *Elastoplasticity Theory*, CRC Press, New York, 2002.

9 Viscoelasticity

9.1 THEORY OF LINEAR VISCOELASTICITY

Elastic materials are those for which stress is proportional to strain. A typical example of elastic behavior is provided by the ideal spring for which force (stress) is proportional to displacement (strain). The ideal spring can be used to model the general behavior of elastic materials in the following manner.

We have seen that an isotropic material is completely characterized by separately giving the constitutive law for volumetric strain and for distortion. For a linear elastic isotropic material, from 2.18,

$$s_{km} = \mu 2 e_{km}, \tag{9.1}$$

$$\frac{1}{3}\tau_{kk} = \kappa \varepsilon_{kk}, \tag{9.2}$$

where

$$\kappa = \lambda + \frac{2}{3}\mu. \tag{9.3}$$

These relations can also be obtained by applying the spring model (Figure 9.1) to each mode of deformation. The distortional strain is $2\boldsymbol{e}$ and the corresponding spring constant is μ. The mean stress is $\tau_{kk}/3$, the volumetric strain is ε_{kk}, and the spring constant for dilatation is κ. The two equations can then be combined by using the definition of the deviatoric components to obtain the single constitutive relation of the linear elastic isotropic material:

$$\tau_{km} = 2\mu\varepsilon_{km} + \lambda\varepsilon_{ii}\delta_{km}. \tag{9.4}$$

Viscous fluids, on the other hand, have a part of the stress that is proportional to the rate of strain. A typical example of a mechanism exhibiting such behavior is the shock absorber or damper mechanism (Figure 9.2), for which force (stress) is proportional to velocity (strain rate).

When the distortion is modeled by a damper, we have in place of 9.1,

$$s_{km} = 2\eta\dot{\mathbf{e}}_{km}, \tag{9.5}$$

where η is called the shear viscosity.

More complicated mechanical behavior is observed in polymers, metals at higher temperatures, and other materials where simultaneous elastic and viscous

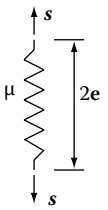


FIGURE 9.1 Spring model.

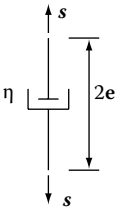


FIGURE 9.2 Damper model.

mechanisms appear in complex ways. Models of such combined viscous and elastic behavior can be developed by combining the spring and damper in series or parallel (Dill: Chapter 5).

The three-element model shown in Figure 9.3 includes the essential characteristics of the behavior of viscoelastic solids. The stress–strain relation corresponding to this model for distortion is found as shown in the following. Equation 9.1 applies to each spring element:

$$\mathbf{s}_1 = 2\mu_1\mathbf{e}_1, \tag{9.6}$$

$$\mathbf{s}_2 = 2\mu_2\mathbf{e}_2. \tag{9.7}$$

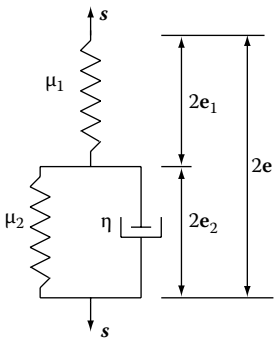


FIGURE 9.3 Standard linear solid.

Equation 9.5 applies to the damper:

$$\mathbf{s}_3 = 2\eta\dot{\mathbf{e}}_3. \tag{9.8}$$

The elements in parallel have a common strain and the total stress is the sum of that on each part:

$$\mathbf{e}_3 = \mathbf{e}_2, \tag{9.9}$$

$$\mathbf{s}_2 + \mathbf{s}_3 = \mathbf{s}. \tag{9.10}$$

The units in series have a common stress and the total strain is the sum of that of each part:

$$\mathbf{s}_1 = \mathbf{s}, \tag{9.11}$$

$$\mathbf{e}_1 + \mathbf{e}_2 = \mathbf{e}. \tag{9.12}$$

The six quantities $\mathbf{s}_1$, $\mathbf{s}_2$, $\mathbf{s}_3$, $\mathbf{e}_1$, $\mathbf{e}_2$, and $\mathbf{e}_3$ are now eliminated from the seven equations, 9.6–9.12, to obtain one stress–strain relation for distortion:

$$\dot{\mathbf{s}} + \frac{\mu_1 + \mu_2}{\eta}\mathbf{s} = 2\mu_1\dot{\mathbf{e}} + \frac{2\mu_1\mu_2}{\eta}\mathbf{e}. \tag{9.13}$$

In addition to the model for distortion, the behavior in dilatation must also be specified. Dilatation is often adequately modeled as elastic so that 9.2 applies. In this case, the complete set of constitutive equations comprises 9.2 and 9.13, which can be rewritten as follows:

$$\tau_{kk} = 3\kappa\varepsilon_{kk}, \tag{9.14}$$

$$\dot{s}_{km} = 2\mu_1\dot{e}_{km} + h_{km}, \tag{9.15}$$

where h_{km} is defined by

$$h_{km} = \frac{2\mu_1\mu_2}{\eta}e_{km} - \frac{\mu_1 + \mu_2}{\eta}s_{km}. \tag{9.16}$$

Equations 9.14 and 9.15 yield the following constitutive relation for the three element model of distortion with elastic volume change:

$$\dot{\tau}_{km} = 2\mu_1\dot{\varepsilon}_{km} + \lambda_1\dot{\varepsilon}_{ii}\delta_{km} + h_{km}, \tag{9.17}$$

where $\lambda_1 = \kappa - 2\mu_1/3$. The material constants occurring in the constitutive relations can be determined by subjecting a rod to a uniaxial stress and measuring both the elongation and the lateral contraction. We will show below that 9.17 holds true for viscoelastic materials in general, but a general expression for h_{km} is needed.

A viscoelastic model based on an assembly of springs and dampers leads to a constitutive equation in the form of a differential equation such as 9.17. To determine the stress at time t for a given history of strain before time t, one must solve the differential equation. The order of the differential equation depends on the number of spring–damper elements to be used to construct the model, but in all cases one is led to a linear differential equation with constant coefficients. The form of the general solution to such an equation is known. For an isotropic material, the general form of the constitutive relation for a material that is unstrained up to $t = 0$ is

$$\tau_{km}(t) = \int_0^t \left\{ 2\mu(t-\tau)\dot{\varepsilon}_{km}(\tau) + \lambda(t-\tau)\dot{\varepsilon}_{ii}(\tau)\delta_{km} \right\} \mathrm{d}\tau. \tag{9.18}$$

Note that we are using the letter τ as a dummy variable of integration and not to denote stress. The context should make the use clear.

The functions $\mu(t)$ and $\lambda(t)$ are called the relaxation moduli and are determined by solving the differential equations provided by the model for a step change in strain at $t = 0$. These functions are given by

$$\mu(t) = A_0 + \sum_{n=1}^{N} A_n \, \mathrm{e}^{-a_n t}, \tag{9.19}$$

$$\begin{aligned} \kappa(t) &= B_0 + \sum_{m=1}^{M} B_m \, \mathrm{e}^{-b_m t}, \\ \lambda(t) &= \kappa(t) - \frac{2}{3}\mu(t), \end{aligned} \tag{9.20}$$

the so-called Prony or Dirichlet series. The constants A_0, A_n, a_n, B_0, B_n, and b_n depend on the stiffness and viscosities of the spring–damper mechanisms. The number of exponential terms is equal to the number of dampers. In practice, the relaxation moduli are directly measured by experimental means for a particular material and the Prony series is numerically fitted to the data.

The constitutive equation 9.18 can be simplified by integration by parts. Note that

$$\frac{\mathrm{d}}{\mathrm{d}\tau}\mu(s(\tau)) = \frac{\mathrm{d}\mu(s)}{\mathrm{d}s}\frac{\mathrm{d}s}{\mathrm{d}\tau} = -\frac{\mathrm{d}\mu(s)}{\mathrm{d}s} = -\dot{\mu}(s) = -\dot{\mu}(t-\tau), \tag{9.21}$$

where $s = t - \tau$. Note that we are using the letter s as a dummy variable of integration and not to denote stress **s**. Therefore,

$$\frac{\mathrm{d}}{\mathrm{d}\tau}\left\{\mu(t-\tau)\varepsilon(\tau)\right\} = \mu(t-\tau)\dot{\varepsilon}(\tau) - \dot{\mu}(t-\tau)\varepsilon(\tau). \tag{9.22}$$

Consequently, integration of 9.18 by parts yields

$$\begin{aligned}\tau_{km}(t) &= 2\mu(0)\varepsilon_{km}(t) + \lambda(0)\varepsilon_{ii}(t)\delta_{km} \\ &\quad + \int_0^t 2\dot{\mu}(t-\tau)\varepsilon(\tau)_{km}\,\mathrm{d}\tau + \int_0^t \dot{\lambda}(t-\tau)\varepsilon_{ii}(\tau)\delta_{km}\,\mathrm{d}\tau,\end{aligned} \tag{9.23}$$

where we have used the initial conditions $\varepsilon_{km}(0) = 0$.

To determine a formula for the incremental stress, let us differentiate 9.23 with respect to t. Recall that

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_0^t f(t,\tau)\,\mathrm{d}\tau = f(t,t) + \int_0^t \frac{\partial f(t,\tau)}{\partial t}\mathrm{d}\tau. \tag{9.24}$$

Thus, the general constitutive equation of an isotropic linearly viscoelastic material is

$$\dot{\tau}_{km}(t) = 2\mu(0)\dot{\varepsilon}_{km}(t) + \lambda(0)\dot{\varepsilon}_{ii}(t)\delta_{km} + h_{km}, \tag{9.25}$$

where

$$\begin{aligned}h_{km} &= 2\dot{\mu}(0)\varepsilon_{km}(t) + \dot{\lambda}(0)\varepsilon_{ii}(t)\delta_{km} \\ &\quad + \int_0^t 2\ddot{\mu}(t-\tau)\varepsilon_{km}(\tau)\,\mathrm{d}\tau + \int_0^t \ddot{\lambda}(t-\tau)\varepsilon_{ii}(\tau)\delta_{km}\,\mathrm{d}\tau.\end{aligned} \tag{9.26}$$

The general equation 9.25 has the same form as 9.17, but h_{km} is given by 9.26.

One can see from 9.23 that stress is determined by the history of strain. The essential computational difference between plasticity and viscoelasticity can be seen in the formulas for the rate of change in stress. In plasticity, the tangent modulus depends on the history of strain and the direction of the strain increment, but there is no memory term h_{km}. For viscoelasticity, the tangent modulus is constant but there is a memory term $h_{km}(t)$.

From the point of view of the theory of material behavior, a viscoelastic material is one for which the stress has a memory of the history of strain, but it is a fading memory. Recent strain has a greater effect on current stress than events that occurred at an earlier time. The material exhibits creep and relaxation, and the stress required

to produce a given deformation history depends on the rate of straining. Plasticity theory, on the other hand, is rate independent. The theory of plasticity applies mostly to metals, whereas viscoelasticity applies mostly to plastics.

9.1.1 Recurrence Formula for History

Let us consider the general constitutive equation of linear viscoelasticity in the form 9.25. The history term $h_{km}(t)$ given in 9.26 depends on the entire history of strain. For numerical calculations, we can use a numerical integration scheme. A simple scheme is diagrammed in Figure 9.4.

Using the illustrated scheme, we have

$$\int_0^t f(\tau)\,\mathrm{d}\tau = \sum_{n=1}^{N} f(t_n)\Delta t_n. \tag{9.27}$$

Applied to 9.26, we have

$$\begin{aligned} h_{km} &= 2\dot{\mu}_0\varepsilon_{km} + \sum_{n=1}^{N} 2\ddot{\mu}(t_N - t_n)\varepsilon_{km}(t_n)\Delta t_n \\ &+ \dot{\lambda}_0\varepsilon_{ii}\delta_{km} + \sum_{n=1}^{N} \ddot{\lambda}(t_N - t_n)\varepsilon_{ii}(t_n)\Delta t_n\delta_{km}. \end{aligned} \tag{9.28}$$

Now, consider the usual case when viscoelastic characteristics are modeled by a finite number of springs and dampers so that Equations 9.19 and 9.20 apply:

$$\begin{aligned} \ddot{\mu}(t_N - t_n) &= \sum_{m=1}^{M} a_m^2 A_m \mathrm{e}^{-a_m t_N}\mathrm{e}^{a_m t_n}, \\ \ddot{\lambda}(t_N - t_n) &= \sum_{m=1}^{M} b_m^2 B_m \mathrm{e}^{-b_m t_N}\mathrm{e}^{b_m t_n}. \end{aligned} \tag{9.29}$$

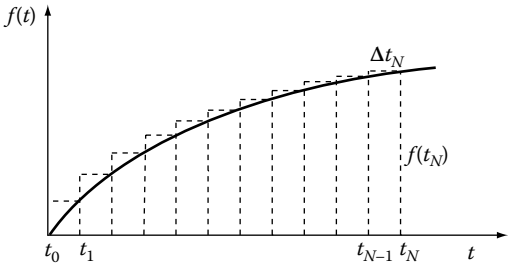


FIGURE 9.4 Numerical integration scheme.

Substituting these expressions into 9.28 gives

$$h_{km} = 2\dot{\mu}_0 \varepsilon_{km} + \sum_{p=1}^{M} a_p^2 A_p \alpha_{kmp} + \dot{\lambda}_0 \varepsilon_{ii} \delta_{km} + \sum_{p=1}^{M} b_p^2 B_p \beta_p \delta_{km}, \tag{9.30}$$

where, at time $t = t_N$,

$$\alpha_{kmp} = 2\mathrm{e}^{-a_p t_N} \sum_{n=1}^{N} \mathrm{e}^{a_p t_n} \varepsilon_{km}(t_n) \Delta t_n,$$
$$\beta_p = \mathrm{e}^{-b_p t_N} \sum_{n=1}^{N} \mathrm{e}^{b_p t_n} \varepsilon_{ii}(t_n) \Delta t_n. \tag{9.31}$$

It follows from 9.31 that $\alpha_{kmp}(t_N)$ and $\beta_p(t_N)$ may be calculated by the following recurrence formulas.

$$\alpha_{kmp}(t_N) = 2\varepsilon_{km}(t_N)\Delta t_N + \mathrm{e}^{-a_p \Delta t_N} \alpha_{kmp}(t_{N-1}),$$
$$\beta_p(t_N) = \varepsilon_{ii}(t_N)\Delta t_N + \mathrm{e}^{-b_p \Delta t_N} \beta_p(t_{N-1}), \tag{9.32}$$

with

$$\alpha_{kmp}(0) = 0,$$
$$\beta_p(0) = 0. \tag{9.33}$$

We have seen that the term h_{km} depends on the strains at all past times. However, it can now be seen from 9.30 to 9.33 that h_{km} can be calculated if the strain at the present time and the immediately previous values of α_{kmp} and β_p are saved in storage. Since the number of dampers M, which are needed to obtain a practical description of a material, will be much smaller than the number N of time steps, there is a considerable saving of computer memory for storage of history data.

9.1.2 Viscoelastic Example

The plane strain problem shown in Figure 9.5 will be solved numerically. The constant load is applied suddenly at time $t = 0$. The three-element model shown in Figure 9.3 will be used for distortion and elastic volume change will be used to

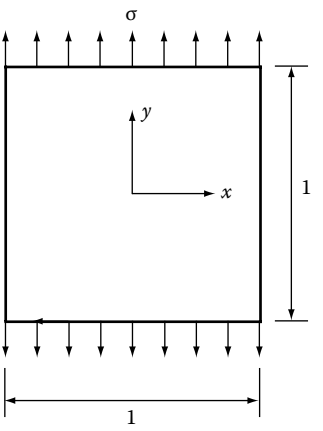


FIGURE 9.5 Plane strain example.

model the material for this example. The constitutive relation consists of 9.14 and 9.15, where μ_1, μ_2, η, and κ have to be assigned numerical values. The exact solution can be easily determined. By inspection, one can guess that the following trial solution may work:

$$\tau_{11} = 0, \quad \tau_{22} \equiv \sigma, \quad \tau_{12} = 0, \quad \tau_{33} \equiv \tau(t), \tag{9.34}$$

where σ is constant and τ is a function of time to be determined. The deviatoric components of the stress tensor are therefore

$$\begin{aligned} s_{11} &= -\frac{1}{3}(\sigma + \tau), \quad s_{22} = \frac{2}{3}\sigma - \frac{1}{3}\tau, \\ s_{12} &= 0, \quad s_{33} = \frac{2}{3}\tau - \frac{1}{3}\sigma. \end{aligned} \tag{9.35}$$

Since $\varepsilon_{33} = 0$ for plane strain, the deviatoric components of the strain tensor are:

$$\begin{aligned} e_{11} &= \frac{2}{3}\varepsilon_{11} - \frac{1}{3}\varepsilon_{22}, \\ e_{22} &= \frac{2}{3}\varepsilon_{22} - \frac{1}{3}\varepsilon_{11}, \\ e_{33} &= -\frac{1}{3}(\varepsilon_{11} + \varepsilon_{22}). \end{aligned} \tag{9.36}$$

A constant $\nu = (3\kappa - 2\mu_1)/(2\mu_1 + 6\kappa)$ analogous to the Poisson ratio of elasticity can be introduced to obtain

$$\tau_{kk} = \frac{2\mu_1(1+\nu)}{(1-2\nu)}\varepsilon_{kk}. \tag{9.37}$$

Since $\varepsilon_{33} = 0$, using 9.34, this relation becomes

$$\varepsilon_{11} + \varepsilon_{22} = \frac{(1-2\nu)}{2\mu_1(1+\nu)}(\sigma + \tau). \tag{9.38}$$

Next, set k and m equal to 3 in the constitutive relation 9.15 for distortion, obtaining

$$\dot{s}_{33} = 2\mu_1\dot{e}_{33} - \frac{\mu_1 + \mu_2}{\eta}s_{33} + \frac{2\mu_1\mu_2}{\eta}e_{33}. \tag{9.39}$$

Using 9.35, 9.36, and 9.38, one finds for $\dot{\sigma} = 0$ that

$$\dot{\tau} + A\tau = B\sigma, \tag{9.40}$$

where

$$A = \frac{1}{3\eta}\Big[2\mu_1(1+\nu) + 3\mu_2\Big], \tag{9.41}$$

$$B = \frac{1}{3\eta}\Big[\mu_1(1+\nu) + 3\nu\mu_2\Big]. \tag{9.42}$$

The solution of 9.40 is

$$\tau(t) = \left(\tau(0) - \frac{B}{A}\sigma\right)\mathrm{e}^{-At} + \frac{B}{A}\sigma. \tag{9.43}$$

Now, return to the relations 9.13, from which

$$\dot{s}_{11} - \dot{s}_{22} = \frac{\mu_1 + \mu_2}{\eta}(s_{22} - s_{11}) + 2\mu_1(\dot{e}_{11} - \dot{e}_{22}) + \frac{2\mu_1\mu_2}{\eta}(e_{11} - e_{22}). \tag{9.44}$$

Using 9.35, 9.36, and 9.38 with $\dot{\sigma} = 0$, one finds

$$\dot{\varepsilon}_{11} - \dot{\varepsilon}_{22} + \frac{\mu_2}{\eta}(\varepsilon_{11} - \varepsilon_{22}) = -\frac{\mu_1 + \mu_2}{2\mu_1\eta}\sigma, \tag{9.45}$$

The solution of this differential equation is

$$\varepsilon_{11}-\varepsilon_{22}=\left[\varepsilon_{11}(0)-\varepsilon_{22}(0)+\frac{\mu_1+\mu_2}{2\mu_1\mu_2}\sigma\right]\mathrm{e}^{-\mu_2 t/\eta}-\frac{\mu_1+\mu_2}{2\mu_1\mu_2}\sigma. \tag{9.46}$$

From 9.38 and 9.46,

$$\begin{aligned}\varepsilon_{22}(t)=&\frac{1-2\nu}{4\mu_1(1+\nu)}\tau+\left[\frac{1-2\nu}{1+\nu}+1+\frac{\mu_1}{\mu_2}\right]\frac{\sigma}{4\mu_1}\\&-\frac{1}{2}\left[\varepsilon_{11}(0)-\varepsilon_{22}(0)+\frac{\mu_1+\mu_2}{2\mu_1\mu_2}\sigma\right]\mathrm{e}^{-\mu_2 t/\eta}.\end{aligned} \tag{9.47}$$

The initial values of strain still have to be calculated. The load is applied suddenly, with a jump in stress at $t = 0$, but no jump in the strain of the damper is allowed. For the initial jump, 9.17 gives

$$\tau_{km}(0)=2\mu_1\varepsilon_{km}(0)+\lambda_1\varepsilon_{ii}(0)\delta_{km}, \tag{9.48}$$

or, with $\varepsilon_{33} = 0$,

$$2\mu_1\varepsilon_{\alpha\beta}(0)=\tau_{\alpha\beta}(0)-\nu\tau_{\gamma\gamma}(0)\delta_{\alpha\beta}. \tag{9.49}$$

Using 9.38, one finds

$$\begin{aligned}2\mu_1\varepsilon_{11}(0)&=-\nu\sigma,\\2\mu_1\varepsilon_{22}(0)&=(1-\nu)\sigma,\\\tau(0)&=\frac{\lambda_1(1-2\nu)}{2\mu_1}\sigma=\nu\sigma.\end{aligned} \tag{9.50}$$

Using these values for the initial conditions, from 9.43,

$$\frac{\tau(t)}{\sigma}=\alpha-(\alpha-\nu)\,\mathrm{e}^{-\beta\bar{t}}, \tag{9.51}$$

and from 9.47,

$$\frac{\mu_1\varepsilon_{22}(t)}{\sigma}=\frac{1-2\nu+(1+\nu)k}{4(1+\nu)}+\frac{(1-2\nu)}{4(1+\nu)}\frac{\tau(t)}{\sigma}-\frac{(1+k-\nu)}{4}\mathrm{e}^{-\bar{t}}, \tag{9.52}$$

TABLE 9.1
Exact Solution

$\bar{t}$	$\bar{u}$
0	0.35
1	0.51407
2	0.57314
3	0.59467
4	0.60256
5	0.60545

where the nondimensional parameters are

$$k = \frac{\mu_1}{\mu_2},\ \alpha = \frac{(1+\nu)k + 3\nu}{2(1+\nu)k + 3},\ \beta = \frac{2(1+\nu)k + 3}{3},\ \bar{t} = \frac{\mu_2 t}{\eta}. \tag{9.53}$$

For the numerical calculations, $k = 1$ and $\nu = 0.3$ will be used. In this case, Equations 9.51 and 9.52 reduce to

$$\frac{\tau(t)}{\sigma} = 0.39286 - 0.09286\, e^{-1.867\bar{t}}, \tag{9.54}$$

$$\bar{u} \equiv \frac{\mu_1 \varepsilon_{22}(t)}{\sigma} = 0.60714 - 0.25\, e^{-\bar{t}} - 0.00714\, e^{-1.867\bar{t}}. \tag{9.55}$$

Table 9.1 shows the exact values of the nondimensional displacement $\bar{u}$ from 9.55 versus the nondimensional time $\bar{t}$.

We will use this as a test problem for the finite element program. Note that the strain is uniform in space, so one 4-node element will yield an exact solution for the spatial distribution, but we must proceed step by step in time because of the creep of the material.

9.2 FINITE ELEMENT FORMULATION FOR VISCOELASTICITY

The fundamental equations, 8.37–8.39, resulting from the balance of internal node forces with external node forces, as introduced in the formulation for plasticity, are true for all materials:

$$\mathbf{f}(t) = \mathbf{F}(t), \tag{9.56}$$

$$\mathbf{f} = \sum_n \int_{V_n} \mathbf{A}^T \boldsymbol{\tau}\, dV. \tag{9.57}$$

$\mathbf{F}(t)$ is the matrix of given external node forces and reactions. The only difference in the formulation for various different materials is the constitutive relation for stress as a function of the history of strain. The constitutive equation (9.25) of a viscoelastic material, in matrix form, is

$$\dot{\boldsymbol{\tau}} = \mathbf{C}\dot{\boldsymbol{\varepsilon}} + \mathbf{h}, \tag{9.58}$$

where $\boldsymbol{\tau}$, $\boldsymbol{\varepsilon}$, and $\mathbf{h}$ denote column matrices, $\mathbf{C}$ is the matrix of instantaneous elasticity, and $\mathbf{h}$ is the memory term. For the plane problem,

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{11} \\ \tau_{22} \\ \tau_{12} \end{bmatrix}, \qquad \boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ 2\varepsilon_{12} \end{bmatrix}, \qquad \mathbf{h} = \begin{bmatrix} h_{11} \\ h_{22} \\ h_{12} \end{bmatrix}. \tag{9.59}$$

The coefficient matrix $\mathbf{C}$ is the matrix of elastic constants with μ and λ replaced by $\mu(0)$ and $\lambda(0)$, respectively. The matrix $\mathbf{h}$ is determined from 9.26 in general, or from 9.16 when the three-element model with elastic volume change is used. In the latter case,

$$\begin{bmatrix} h_{11} \\ h_{22} \\ h_{12} \end{bmatrix} = \frac{2\mu_1\mu_2}{\eta} \begin{bmatrix} e_{11} \\ e_{22} \\ e_{12} \end{bmatrix} - \frac{\mu_1 + \mu_2}{\eta} \begin{bmatrix} s_{11} \\ s_{22} \\ s_{12} \end{bmatrix}. \tag{9.60}$$

The elastic volume change 9.14 is used to find τ_{33} in plane strain or ε_{33} in the case of plane stress.

9.2.1 Basic Step-by-Step Solution Method

From 8.35, 9.57, and 9.58,

$$\dot{\mathbf{f}}(t) = \mathbf{K}\dot{\mathbf{D}}(t) + \mathbf{H}(t), \tag{9.61}$$

where

$$\mathbf{K} = \sum_n \int_{\mathbf{V}_n} \mathbf{A}^{\mathrm{T}}\mathbf{C}\mathbf{A}\, \mathrm{d}V \tag{9.62}$$

is the stiffness matrix for the instantaneous elasticity, and

$$\mathbf{H}(t) = \sum_n \int_{\mathbf{V}_n} \mathbf{A}^{\mathrm{T}}\mathbf{h}(t)\, \mathrm{d}V \tag{9.63}$$

is the memory load. The balance of external and internal nodal forces 9.56 leads to the equation for $\dot{\mathbf{D}}$ as follows:

$$\mathbf{K}\dot{\mathbf{D}}(t) = \dot{\mathbf{F}}(t) - \mathbf{H}(t). \tag{9.64}$$

Note that the stiffness matrix is constant for viscoelastic materials, but the right-hand side contains the memory load as well as the external nodal forces. We can now proceed in a step-by-step manner to construct the solution as a function of time. The fundamental procedure is to use the lower-order terms of the power series representation to march the solution forward. That is, having determined the solution up to time t_n, so that $\mathbf{D}(t_n)$ and $\tau(t_n)$ are known, the solution at time t_{n+1} is determined by

$$\mathbf{D}(t_{n+1}) = \mathbf{D}(t_n) + \dot{\mathbf{D}}(t_n)\Delta t, \tag{9.65}$$

$$\boldsymbol{\tau}(t_{n+1}) = \boldsymbol{\tau}(t_n) + \dot{\boldsymbol{\tau}}(t_n)\Delta t. \tag{9.66}$$

Any finite element code for thermoelastic stress analysis is easily extended to assemble and solve Equation 9.64. Note that $\mathbf{K}$ needs be formed and inverted only once. Equation 9.64 forms a set of linear ordinary differential equations in time. The solution procedure can be improved by using the solution methods for dynamical problems, which will be discussed in the next chapter.

9.2.2 Step-by-Step Calculation with Load Correction

The basic step-by-step method can be improved if we take into account that $\mathbf{f}(t_n)$ is only approximately equal to $\mathbf{F}(t_n)$. The incremental load then consists of the memory load and the unbalanced node forces as follows. Having determined the solution up to time t, we seek the solution at time $t + \mathrm{d}t$ for which

$$\mathbf{f}(t + \Delta t) = \sum_n \int_{\mathbf{V}_n} \mathbf{A}^{\mathrm{T}} \boldsymbol{\tau}(t + \Delta t)\, \mathrm{d}V. \tag{9.67}$$

Writing $\tau(t + \Delta t) = \tau(t) + \Delta\tau$, we have

$$\mathbf{f}(t + \Delta t) = \mathbf{f}(t) + \sum_n \int_{\mathbf{V}_n} \mathbf{A}^{\mathrm{T}} \Delta\boldsymbol{\tau}\, \mathrm{d}V. \tag{9.68}$$

Since

$$\Delta\tau = \mathbf{C}\Delta\varepsilon + \mathbf{h}\Delta t \tag{9.69}$$

and

$$\Delta\varepsilon = \mathbf{A}\Delta\mathrm{D}, \tag{9.70}$$

we have

$$\mathbf{f}(t + \Delta t) = \mathbf{f}(t) + \mathbf{K}\Delta\mathrm{D} + \Delta\mathrm{H}, \tag{9.71}$$

where

$$\Delta\mathbf{H} = \sum_{n} \int_{V_n} \mathbf{A}^{\mathrm{T}}\mathbf{h}\Delta t \, \mathrm{d}V. \tag{9.72}$$

We seek a solution for which $\mathbf{f}(t + \Delta t) = \mathbf{F}(t + \Delta t)$. Therefore, 9.71 becomes

$$\mathbf{K}\Delta\mathbf{D}(t) = \mathbf{F}(t + \Delta t) - \mathbf{f}(t) - \Delta\mathrm{H}(t). \tag{9.73}$$

The finite element program is used to solve for $\Delta\mathbf{D}$ and the solution is marched forward as before.

9.2.3 Plane Strain Example

The example problem, for which the exact solution is given in Section 9.1.2, was solved using one 4-node square element. Since the strains are uniform in the exact solution, no approximation is introduced by the finite element idealization and one element provides an exact representation. The only error here is due to the numerical integration in time. The results using the basic step-by-step formula 9.64 are shown in Figure 9.6. As the exact solution 9.55 shows, the solution to this particular problem

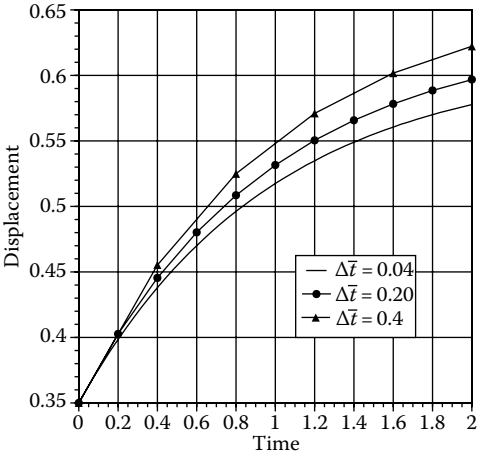


FIGURE 9.6 Basic step-by-step solution.

TABLE 9.2
Step-by-Step Solution

$\bar{t}$	$\bar{u}$ for $\Delta\bar{t} = 0.40$	$\bar{u}$ for $\Delta\bar{t} = 0.04$
0	3.500	3.500
0.40	4.353	4.362
0.80	4.920	4.932
1.20	5.299	5.311
1.60	5.552	5.563
2.00	5.722	5.731

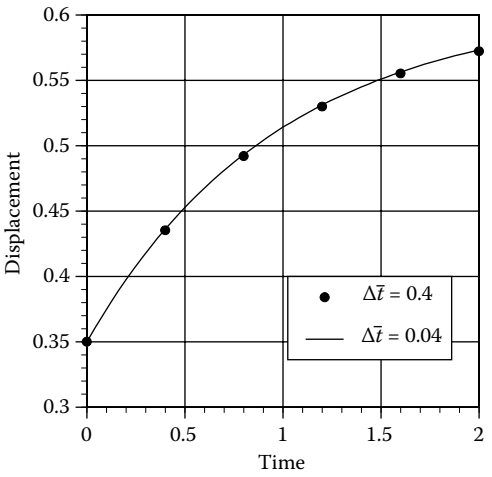


FIGURE 9.7 Solution with load correction.

depends only on the nondimensional time $\bar{t} = \mu_2 t/\eta$. The numerical results for the nondimensional displacement $\bar{u}$ are plotted for three step sizes in $\bar{t}$. The exact solution coincides with the results for $d\bar{t} = 0.04$ up to the resolution of the graph.

The necessary size of time step is greatly reduced if we use the step-by-step calculation with the load correction 9.73. The results for $\Delta\bar{t} = 0.4$ and $\Delta\bar{t} = 0.04$ agree to two significant figures. Typical calculated values are shown in Table 9.2 and plotted in Figure 9.7.

9.3 PROBLEMS

1. Consider a two element Kelvin–Voigt model for both shear and volume change (see Figure 9.8).
   (a) Show that $s_{ij} = 2\mu e_{ij} + 2\eta\dot{e}_{ij}$ and $\frac{1}{3}\tau_{kk} = \kappa\varepsilon_{kk} + \zeta\dot{\varepsilon}_{kk}$.

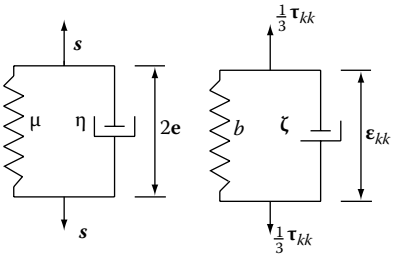


FIGURE 9.8 Kelvin–Voigt model.

(b) Show that these equations are equivalent to the single constitutive equation $\tau_{km} = 2\mu\varepsilon_{km} + \lambda\varepsilon_{ii}\delta_{km} + 2\eta\dot{\varepsilon}_{km} + \lambda_D\dot{\varepsilon}_{ii}\delta_{km}$ where $\lambda = b - \frac{2}{3}\mu$, $\lambda_D = \zeta - \frac{2}{3}\eta$.

2. Given the standard linear solid model for distortion (Figure 9.3), determine the creep compliance.

   Answer: $$\frac{2e_{12}(t)}{s_{12}} = \left(\frac{1}{\mu_1} + \frac{1}{\mu_2} - \frac{1}{\mu_2}\exp(-\mu_2 t/\eta)\right)$$

3. In matrix form, the constitutive relation for the Kelvin–Voigt material is $\boldsymbol{\tau} = \mathbf{C}\boldsymbol{\varepsilon} + \mathbf{C}_D\dot{\boldsymbol{\varepsilon}}$.

   Derive the finite element form of the equilibrium equations for this material model.

4. Determine the creep compliance for the tensile test of a material modeled by an elastic volume change and the three-element model in shear.

5. Analyze the creep test by the finite element method from the given relaxation modulus and compare with the results of problem 5. See Section 15.14.

6. Solve the example illustrated in Figure 9.5 using ANSYS (see Section 15.15). Use $G(0) = 100$, $G(\infty) = 50$, $K(0) = K(\infty) = 650/3$. There is one dashpot with $G_1 = G(0) - G(\infty)$ and a relaxation time of 0.5. The load $\sigma = 1$ is constant (step) in time. Submit:
   (a) A list of UY at the corner for each time step $0 < t \leq 3$.
   (b) A plot of UY(t).

BIBLIOGRAPHY

Bland, D. R., *The Theory of Linear Viscoelasticity*, Pergamon Press, Oxford, England, 1960.

Christensen, R. M., *Theory of Viscoelasticity*, 2nd ed., Academic Press, New York, 1982.

Leitman, M. J., and G. M. Fisher, The linear theory of viscoelasticity, in *Encyclopedia of Physics*, Vol. VIa/3, ed. S. Flügge, Springer-Verlag, New York, 1973.

Lockett, F. J., *Nonlinear Viscoelastic Solids*, Academic Press, New York, 1972.

10 Dynamic Analyses

10.1 DYNAMICAL EQUATIONS

In this section, we shall formulate finite element equations that include the inertial forces that have been neglected up to now.

10.1.1 Lumped Mass

The most direct way of discovering suitable equations governing motion is to concentrate the total mass at node points and apply d'Alembert's principle. Consider, for example, the solution of the plane problem using the rectangular elements (Section 5.1.4). A typical node arrangement is shown in Figure 10.1. The total mass of the material surrounding a node, shown by dashed lines, is treated as a point mass at that node.

Suppose the node spacing is uniformly "a by b" and the mass density of a unit slice of material is ρ. At an interior point, the lumped mass is ρab. At a corner the lumped mass is 1/4 ρab, and at a side point the lumped mass is 1/2 ρab. The effect of inertia is then to exert at each node a force "$-m\ddot{\mathbf{D}}$". At an interior node, for example, the x and y components of inertial force are $(-\rho ab\ddot{U}, -\rho ab\ddot{V})$. For the entire body, the inertial forces have the form

$$\mathbf{F}_I = -\mathbf{M}\ddot{\mathbf{D}}, \tag{10.1}$$

where $\mathbf{M}$ is a diagonal matrix continuing the list of lumped masses as shown in Figure 10.1. The complete equations of motion are then obtained from static formulation by adding these inertial forces to any applied external loads:

$$\mathbf{KD} = \mathbf{F} - \mathbf{M}\ddot{\mathbf{D}}, \tag{10.2}$$

where $\mathbf{F}$ denotes the matrix of external loads. Recall that rigid translation and rotation are allowed by 10.2 if no boundary conditions on displacement have been imposed.

Equation 10.2 has the form of the usual dynamical equations of discrete systems. We may think of Equation 10.2 as representing N masses connected by "springs" and proceed to solve them in the usual fashion. From that point of view, the finite element procedure merely determines the effective spring constants.

In the lumped mass approach, the finite element idealization is not directly connected with the dynamical idealization, other than to indicate an appropriate set of points for the lumped masses. The distribution of mass between the nodes, as just described, is a rather arbitrary choice, but it has been found to work well in practice as long as all mass is accounted for. However, one can proceed in a more rational manner, as we shall now present.

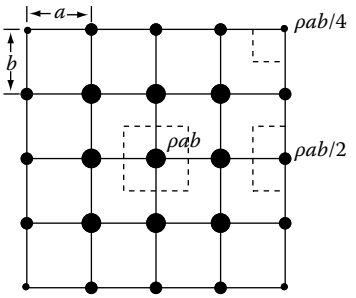


FIGURE 10.1 Lumped mass.

10.1.2 Consistent Mass

The virtual work theorem applies to the dynamic problem if the body force includes the inertial force. Writing **b** for the body force excluding the inertia, we have

$$\int_S T_i \bar{u}_i \, dA + \int_V b_i \bar{u}_i \, dV - \int_V \rho \ddot{u}_i \bar{u}_i \, dV = \int_V \tau_{ij} \bar{\varepsilon}_{ij} \, dV. \tag{10.3}$$

The $\bar{u}_i$ are arbitrary continuous functions and $\bar{\varepsilon}_{ij} = \bar{u}_{(i,j)}$. The shape functions of the finite element method are again used to generate the virtual displacement field:

$$\mathbf{u} = \mathbf{N}\mathbf{D}, \quad \bar{\mathbf{u}} = \mathbf{N}\bar{\mathbf{D}}. \tag{10.4}$$

That is, the shape functions **N** are used to generate the functions $\bar{\mathbf{u}}$ from arbitrary nodal displacements $\bar{\mathbf{D}}$. Substituting 10.4 into 10.3, which must be identically true for all $\bar{\mathbf{D}}$, introduces a new term in the equilibrium equations—the inertial force:

$$\mathbf{F}_I = -\mathbf{M}\ddot{\mathbf{D}} \tag{10.5}$$

as for the lumped mass model, but we now have a formula for the mass matrix:

$$\mathbf{M} = \sum_n \int_{V_n} \rho \mathbf{N}^T \mathbf{N} \, dV, \tag{10.6}$$

where **M** is called the consistent mass.

As an example, consider the rod element of length L, area A, and density ρ. The shape functions are

$$N_1 = \frac{x_2 - x}{L}, \quad N_2 = \frac{x - x_1}{L}. \tag{10.7}$$

The consistent mass matrix is

$$\mathbf{M} = \int_{x_1}^{x_2} \rho \begin{bmatrix} N_1 \\ N_2 \end{bmatrix} \begin{bmatrix} N_1 & N_2 \end{bmatrix} A \mathrm{d}x = \frac{\rho AL}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}. \tag{10.8}$$

Let us next consider the rectangular element described in Section 5.1. If we order the list of nodes by grouping all x components followed by all y components, $\mathbf{D}^{\mathrm{T}} = [\mathbf{U}^{\mathrm{T}},\mathbf{V}^{\mathrm{T}}]$, the shape functions are

$$\begin{matrix} & \mathbf{U} & \mathbf{V} \end{matrix}$$
$$\mathbf{N} = \begin{bmatrix} \lfloor N_k \rfloor & \mathbf{0} \\ \mathbf{0} & \lfloor N_k \rfloor \end{bmatrix}, \quad N_k = \frac{1}{4}(1+\xi_k\xi)(1+\eta_k\eta). \tag{10.9}$$

Using 10.6, one finds

$$\begin{matrix} & \mathbf{U} & \mathbf{V} \end{matrix}$$
$$\mathbf{M} = \begin{bmatrix} \mathbf{M}_1 & 0 \\ 0 & \mathbf{M}_1 \end{bmatrix}, \tag{10.10}$$

where

$$\begin{aligned} \mathbf{M}_1 &= \iint \mathbf{N}^{\mathrm{T}}\mathbf{N}\rho \,\mathrm{d}A \\ &= \frac{ab}{4} \int_{-1}^{+1}\int_{-1}^{+1} \mathbf{N}^{\mathrm{T}}\mathbf{N}\rho \,\mathrm{d}\zeta \,\mathrm{d}\eta \\ &= \frac{\rho ab}{36} \begin{bmatrix} 4 & & & \\ 2 & 4 & & \mathrm{sym} \\ 1 & 2 & 4 & \\ 2 & 1 & 2 & 4 \end{bmatrix}. \end{aligned} \tag{10.11}$$

As a third example, let us consider the triangular element discussed in Section 4.2. Again, grouping the nodes by component, the shape functions from 4.16 are

$$\mathbf{N} = \begin{bmatrix} \zeta_1 & \zeta_2 & \zeta_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & \zeta_1 & \zeta_2 & \zeta_3 \end{bmatrix}, \tag{10.12}$$

where ζ_k are the triangular coordinates defined in Section 4.1. From 10.6, using (4.9), **M** again has the form 10.10, but

$$\mathbf{M}_1 = \rho \frac{A}{12} \begin{bmatrix} 2 & 1 & 1 \\ 1 & 2 & 1 \\ 1 & 1 & 2 \end{bmatrix}, \tag{10.13}$$

where A is the area of the triangle.

In each case, the consistent mass matrix is not diagonal and the off-diagonal terms are rather significant in size. The mass matrices are, however, symmetric and positive definite.

For both mass models, the basic equations finite element with the inertial forces become

$$\mathbf{M}\ddot{\mathbf{D}}(t) + \mathbf{K}\mathbf{D}(t) = \mathbf{F}(t). \tag{10.14}$$

These basic equations, N in number, have the standard form for discrete linear systems with N degrees of freedom. There are two common approaches taken to solve them, either mode superposition or direct integration. Each method will be addressed in the following sections.

10.2 NATURAL FREQUENCIES

The modes of vibration and the natural frequencies of vibration are first found by solving 10.14 for the case of zero external force:

$$\mathbf{M}\ddot{\mathbf{D}}(t) + \mathbf{K}\mathbf{D}(t) = \mathbf{0}. \tag{10.15}$$

The solutions of 10.15 have the form $\mathbf{D}(t) = \mathbf{D}_n \exp(i\omega_n t)$, with no sum on n. Substituting this expression into 10.15 yields the fundamental eigenvalue problem:

$$(\mathbf{K} - \omega_n^2 \mathbf{M})\mathbf{D}_n = 0, \quad \text{(no sum on } n\text{)}. \tag{10.16}$$

The stiffness matrix **K** is symmetric, nonnegative, and banded. The mass matrix **M** is symmetric, positive definite, and banded. If lumped masses are assigned and some inertial degree of freedom is neglected, a singular mass matrix may occur, and due care should be exercised in dealing with these equations in such cases.

10.2.1 Lumped Mass

The lumped mass approach is convenient since **M** is then diagonal, and $\mathbf{M}^{1/2}$ is also diagonal. In this case, let

$$\mathbf{X}_n = \mathbf{M}^{1/2}\mathbf{D}_n, \tag{10.17}$$

$$\mathbf{A} = \mathbf{M}^{-1/2}\mathbf{K}\mathbf{M}^{-1/2}. \tag{10.18}$$

The eigenvalue problem 10.16 becomes

$$(\mathbf{A} - \omega_n^2 \mathbf{I})\mathbf{X}_n = 0, \tag{10.19}$$

with **A** symmetric. This equation is a standard problem in numerical analysis, and many effective solution algorithms are available. Once solution vectors $\mathbf{X}_n$ are known, the mode shapes are calculated by 10.17: $\mathbf{D}_n = \mathbf{M}^{-1/2}\mathbf{X}_n$.

10.2.2 Consistent Mass

In the general case, when **M** is not a diagonal matrix but is symmetric positive definite, we can proceed as follows in order to reduce the eigenvalue problem to the standard form. There exists triangular matrices **L** such that

$$\mathbf{M} = \mathbf{L}\mathbf{L}^{\mathrm{T}}. \tag{10.20}$$

The matrix **L** is easily inverted. There is a standard procedure for calculating **L** known as the Choleski decomposition. Now, define

$$\mathbf{A} = \mathbf{L}^{-1}\mathbf{K}(\mathbf{L}^{-1})^{\mathrm{T}}, \tag{10.21}$$

$$\mathbf{X}_n = \mathbf{L}^{\mathrm{T}}\mathbf{D}_n, \tag{10.22}$$

and the standard eigenvalue problem 10.19 results.

10.3 MODE SUPERPOSITION SOLUTION

It is now assumed that the N solutions of 10.16 have been found. The modal vectors satisfy the orthogonality conditions ($n \neq m$):

$$(\mathbf{D}_n)^{\mathrm{T}}\mathbf{K}\mathbf{D}_m = 0, \tag{10.23}$$

$$(\mathbf{D}_n)^{\mathrm{T}}\mathbf{M}\mathbf{D}_m = 0. \tag{10.24}$$

The solution $\mathbf{D}(t)$ of the complete equations 10.15 can be expressed as a linear combination of the mode shapes:

$$\begin{aligned} &\mathbf{D}(t) = \sum_n \mathbf{D}_n q_n(t) = [\mathbf{T}]\{\mathbf{q}(t)\}, \\ &[\mathbf{T}] = [D_{in}], \quad \{\mathbf{q}(t)\} = \{q_n(t)\}, \end{aligned} \tag{10.25}$$

where **T** is a square matrix whose columns are the mode shape vectors. Substituting 10.25 into 10.15, and using 10.23 and 10.24, one finds the set of uncoupled equations:

$$\ddot{q}_n + \omega_n^2 q_n = \frac{P_n}{M_n}, \quad \text{no sum on } n, \tag{10.26}$$

where

$$\begin{aligned} M_n &= (\mathbf{D}_n)^{\mathrm{T}} \mathbf{M} \mathbf{D}_n, \\ K_n &= (\mathbf{D}_n)^{\mathrm{T}} \mathbf{K} \mathbf{D}_n = \omega_n^2 M_n, \\ P_n &= (\mathbf{D}_n)^{\mathrm{T}} \mathbf{F}. \end{aligned} \tag{10.27}$$

The modal equations 10.26 have the solution

$$q_n = \frac{1}{M_n \omega_n} \int_0^t P_n(\tau) \sin\left[\omega_n (t-\tau)\right] \mathrm{d}\tau = \frac{1}{M_n \omega_n} \int_0^t P_n(t-\tau) \sin\left(\omega_n \tau\right) \mathrm{d}\tau. \tag{10.28}$$

If P_n is constant, that is, a step function at $t = 0$, then

$$q_n = \frac{P_n}{K_n}(1 - \cos(\omega_n t)). \tag{10.29}$$

If P_n is ramped up to a constant value as shown in Figure 10.2, then

$$q_n(t) = \frac{P_0}{K_n}\left(\frac{t}{t_1} - \frac{\sin(\omega t)}{\omega t_1}\right), \quad 0 \le t \le t_1, \tag{10.30}$$

$$q_n(t) = \frac{P_0}{K_n}\left(1 - \frac{\sin(\omega t_1/2)}{\omega t_1/2}\cos(\omega \bar{t})\right), \quad t_1 \le t, \tag{10.31}$$

where $\bar{t} = t - t_1/2$.

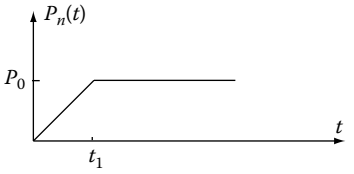


FIGURE 10.2 Ramped load.

10.4 EXAMPLE: AXIALLY LOADED ROD

As an example for which the exact solution can be determined analytically, we consider the uniaxial stress field approximated by an axial load applied to a rod (Figure 10.3).

10.4.1 Exact Solution for Axially Loaded Rod

To understand the finite element solution, it is necessary to understand the relation of the discrete model to the continuum model. Let us illustrate this by constructing the exact solution for the axial deformation of a rod with a step load on the end. The equations of elasticity for uniaxial stress σ and axial displacement u are as follows:

$$\frac{\partial \sigma}{\partial x} = \rho \frac{\partial^2 u}{\partial t^2} \quad \varepsilon = \frac{\partial u}{\partial x}, \quad \sigma = E\varepsilon. \tag{10.32}$$

The initial conditions are

$$u(x,0) = 0, \quad \dot{u}(x,0) = 0. \tag{10.33}$$

The boundary conditions are

$$\sigma(0,t) = -P/A, \quad u(L,t) = 0. \tag{10.34}$$

The basic equations can be combined to obtain one equation for the axial displacement:

$$c^2 \frac{\partial^2 u}{\partial x^2} = \frac{\partial^2 u}{\partial t^2}, \quad c^2 = \frac{E}{\rho}. \tag{10.35}$$

This is the classical wave equation. The general solution is

$$u(x,t) = f(s), \quad s = x \pm ct. \tag{10.36}$$

Each such solution is a wave traveling either to the right or to the left. A complete solution that satisfies the initial conditions and the boundary conditions can be obtained as a linear combination of particular solutions of this type.

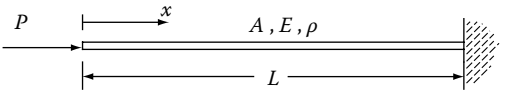


FIGURE 10.3 Axially loaded rod.

For $0 \le ct \le L$, in order to satisfy the initial conditions and the boundary conditions, we must take $s = x - ct$ and

$$u(x,t) = \begin{cases} -\dfrac{P}{AE}s, & \text{for } x \le ct, \\ 0, & \text{for } x > ct. \end{cases} \tag{10.37}$$

$$\sigma(x,t) = \begin{cases} -\dfrac{P}{A}, & \text{for } x \le ct, \\ 0, & \text{for } x > ct. \end{cases} \tag{10.38}$$

For $ct > L$, this solution violates the boundary condition at the fixed end. A second solution must be added that counteracts the first one in order to satisfy the boundary condition at $x = L$. Then, a third solution is added to correct the boundary condition at $x = 0$, and so forth.

Let us introduce nondimensional variables:

$$\bar{x} = \frac{x}{L}, \quad \bar{t} = \frac{ct}{L}, \quad \bar{u} = \frac{AEu}{PL}, \quad \bar{\sigma} = \frac{A\sigma}{P}. \tag{10.39}$$

The complete wave solution is shown in Figures 10.4 and 10.5. Figure 10.4 shows the displacement at the loaded end and the displacement of the midpoint versus time. Figure 10.5 shows the stress and displacement versus x at various times.

When a load is suddenly applied to the end of the rod, it causes a stress wave to propagate along the rod. The wave is reflected from the fixed end, doubling the magnitude of the stress. The wave is again reflected when it returns to the loaded end, reducing the stress to the initial value, and so on. There is no displacement of a particle of the rod until the wave reaches it; then the displacement commences and can be calculated from the known uniform strain of the portion of the rod that has been covered by the wave. The speed of propagation of the wave is $c = \sqrt{E/\rho}$.

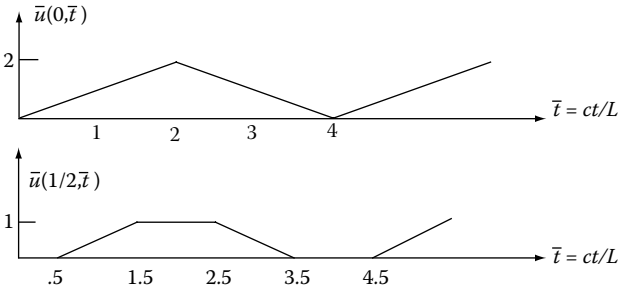


FIGURE 10.4 Exact displacements.

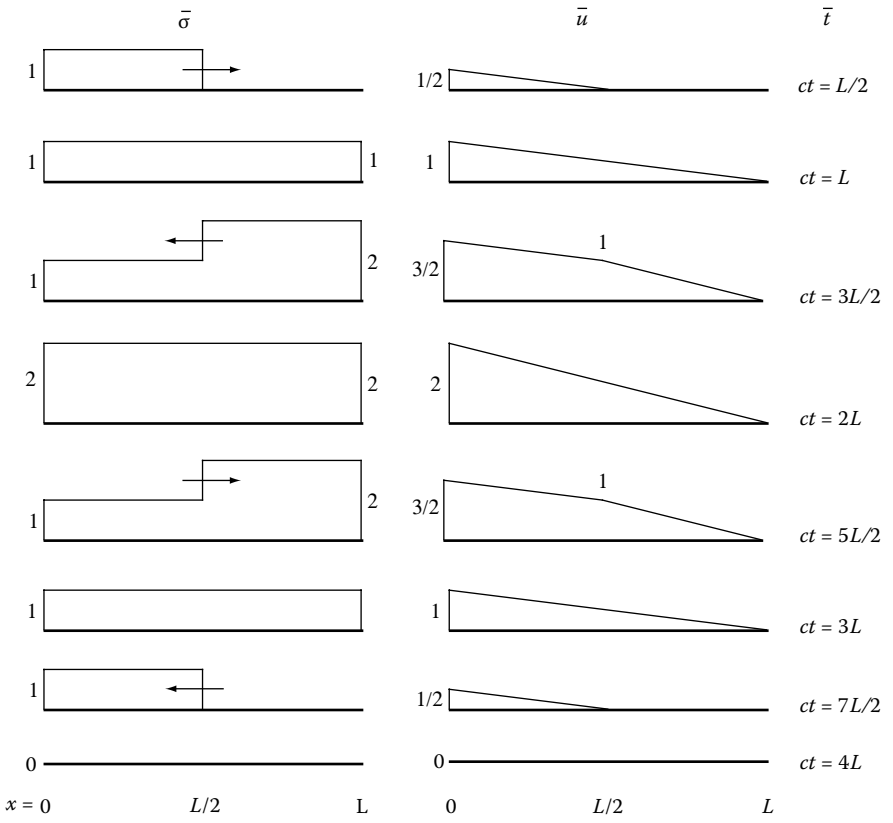


FIGURE 10.5 Exact stress and displacement.

10.4.2 Finite Element Model

10.4.2.1 One-Element Model

For one-element models with a lumped mass at each end, the finite element equation is

$$m\ddot{D}(t) + kD(t) = P(t) \tag{10.40}$$

where

$$m = \frac{\rho AL}{2}, \quad k = \frac{AE}{L}, \quad \omega^2 = \frac{k}{m}. \tag{10.41}$$

For a step load at $t = 0$,

$$D(t) = \frac{P}{k}(1 - \cos(\omega t)). \tag{10.42}$$

For a ramped load (Figure 10.2),

$$D(t) = \frac{P_0}{k}\left(\frac{t}{t_1} - \frac{\sin(\omega t)}{\omega t_1}\right), \quad 0 \le t \le t_1,$$
$$D(t) = \frac{P_0}{k}\left(1 - \frac{\sin(\omega t_1/2)}{\omega t_1/2}\cos(\omega \bar{t})\right), \quad \bar{t} = t - t_1/2, \quad t_1 \le t. \tag{10.43}$$

10.4.2.2 Two-Element Model

Let us analyze the motion of the rod by using a finite element model as shown in Figure 10.6. Two elements are used with a node at the midpoint of the rod.

The lumped mass procedure is used. The equations of motion, which were formulated in Chapter 1, are as follows:

$$\begin{bmatrix} \frac{\rho AL}{4} & 0 \\ 0 & \frac{\rho AL}{2} \end{bmatrix} \begin{bmatrix} \ddot{D}_1 \\ \ddot{D}_2 \end{bmatrix} + \begin{bmatrix} \frac{2AE}{L} & -\frac{2AE}{L} \\ -\frac{2AE}{L} & \frac{4AE}{L} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \end{bmatrix} = \begin{bmatrix} P \\ 0 \end{bmatrix}. \tag{10.44}$$

If a load is suddenly applied at node 1, the complete solution can be found by the mode superposition method. The displacements of the nodes are

$$D_1 = \frac{PL}{2AE}\left(2 - (1 + \frac{1}{\sqrt{2}})\cos\omega_1 t - (1 - \frac{1}{\sqrt{2}})\cos\omega_2 t\right),$$
$$D_2 = \frac{PL}{AE}\sqrt{2}\left(\frac{1 - \cos\omega_1 t}{8 - 4\sqrt{2}} - \frac{1 - \cos\omega_2 t}{8 + 4\sqrt{2}}\right), \tag{10.45}$$

where

$$\omega_1^2 = \left(1 - \frac{1}{\sqrt{2}}\right)\frac{8E}{\rho L^2}, \quad \omega_2^2 = \left(1 + \frac{1}{\sqrt{2}}\right)\frac{8E}{\rho L^2}. \tag{10.46}$$

Unlike the wave solution, there is an immediate displacement of all nodes. The maximum value of D_1 is found to be 1.88 *PL/AE*. That is, the dynamic magnification factor is 1.88 for the two-element model of the rod. The magnification will approach 2 as the number of elements increases.

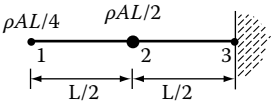


FIGURE 10.6 Two-element model of the rod.

The solution (10.45) for the two-element model is compared with the exact wave solution in Figure 10.7 for $\frac{\rho L^2}{8E} = \frac{L^2}{8c^2} = 1$. The normalized displacement $D(t)/D_0$, $D_0 = \frac{PL}{AE}$, is plotted as a function of nondimensional time $\bar{t} = ct/L$. The value $D(t)/D_0 = 1$ is the static solution.

Figure 10.8 is a plot of $D_2/(PL/AE)$ versus $\bar{t}$ showing both the two-element finite element analysis and the wave solution.

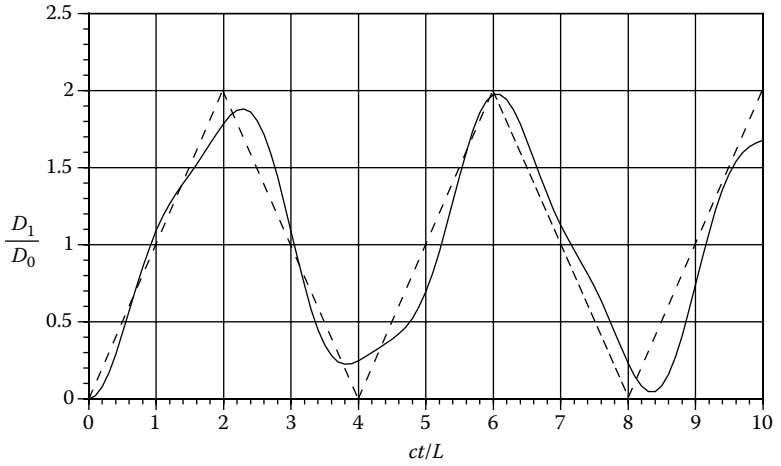


FIGURE 10.7 Exact solution for continuum and two-element model.

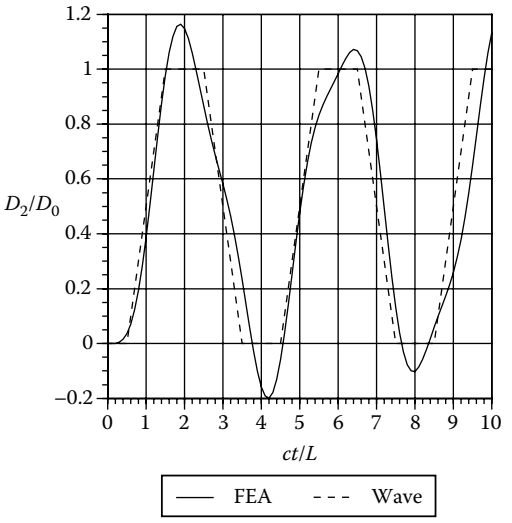


FIGURE 10.8 Displacement of node 2 of two-element model.

The dynamical solution of the continuum model is a wave propagation. There is no motion of a particle until the stress wave reaches it. In the absence of damping, the wave bounces around forever. Damping causes the waves to die out as the motion approaches a steady state.

The finite element model is a discrete system. All nodes are put into motion immediately, as if there was an infinite wave speed. The finite element solution will approximate the wave solution of the continuum problem in a manner similar to a Fourier series representation. We will always obtain a smooth curve from the finite element analysis that is a superposition of the available mode shapes. We will not get a straight line because we are merely adding together a finite number of cosine functions, and there will always be a waviness in the finite element solution corresponding to the higher-order frequencies that are excited by the applied load.

10.4.3 Mode Superposition for Continuum Model of the Rod

The similarity of the finite element solution to the exact solution is more apparent if the exact solution is also expressed by mode superposition. We first determine the infinity of mode shapes and characteristic frequencies of the continuum model of the rod with one end free and the other fixed. The differential equation to be solved is

$$c^2 \frac{\partial^2 u(x,t)}{\partial x^2} = \frac{\partial^2 u(x,t)}{\partial t^2}. \tag{10.47}$$

The boundary conditions are

$$u(0,t) = 0, \quad \left.\frac{\partial u(x,t)}{\partial x}\right|_{x=L} = 0. \tag{10.48}$$

Free vibrations are so-called standing waves of the form

$$u(x,t) = U(x)(A\cos \omega t + B\sin \omega t). \tag{10.49}$$

Substitution of this function into the differential equation yields a solution if

$$c^2 \frac{\mathrm{d}^2 U(x)}{\mathrm{d}x^2} + \omega^2 U(x) = 0. \tag{10.50}$$

The general solution to this differential equation is

$$U(x) = C\cos\frac{\omega x}{c} + D\sin\frac{\omega x}{c}. \tag{10.51}$$

The boundary conditions 10.48 require that $C = 0$ and

$$\cos\frac{\omega L}{c} = 0. \tag{10.52}$$

This yields the characteristic frequencies:

$$\omega = \frac{n\pi c}{2L}, \quad n \text{ odd.} \tag{10.53}$$

The family of mode shapes is therefore

$$U_n(x) = \sin \frac{\omega_n x}{c}. \tag{10.54}$$

This exact solution for the natural frequencies 10.53 is compared to the approximate solution obtained by the finite element analysis using the lumped mass and consistent mass in Table 10.1. Convergence toward the first frequency is shown in

TABLE 10.1
Finite Element Analysis

Number of Elements	Exact $2\pi\omega_1 L/c$	FEA Lumped Mass	FEA Consistent Mass
1	0.25	0.22508	0.27566
2	0.25	0.243624	0.256465
	0.75	0.588160	0.895931
3	0.25	0.247154	0.252865
	0.75	0.675237	0.826993
	1.25	0.922391	1.50029
4	0.25	0.248397	0.251609
	0.75	0.707374	0.793737
	1.25	1.05886	1.44185
	1.75	1.24877	2.0804
5	0.25	0.248973	0.251029
	0.75	0.722548	0.777971
	1.25	1.12540	1.37832
	1.75	1.41808	2.06686
	2.25	1.57195	2.65843
10	0.25	0.249753	0.250257
	0.75	0.743080	0.756957
	1.25	1.21812	1.28232
	1.75	1.66316	1.83891
	2.25	2.06726	2.43830
100	0.25	0.249997	0.250003
	0.75	0.749931	0.750069
	1.25	1.24968	1.25032
	1.75	1.74912	1.75088
	2.25	2.24813	2.25187

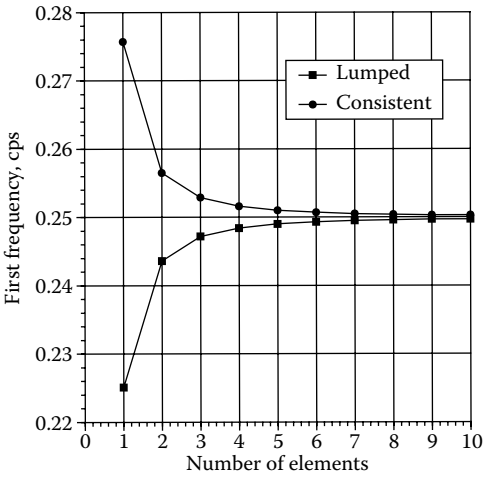


FIGURE 10.9 First Frequency by FEM.

Figure 10.9. The lumped mass converges from below and the consistent mass converges from above. The dynamic magnification factor also depends on the number of elements.

The deformations resulting from a step load applied to the free end can be represented by an infinite series using the mode shapes and natural frequencies:

$$u(x,t) = \sum_{n\,\text{odd}} q_n(t) \sin \frac{\omega_n x}{c}\,. \tag{10.55}$$

The modal amplitudes can be determined by using virtual work to express the equilibrium conditions:

$$\int_0^L \sigma\,\bar{\varepsilon}\,A\,\mathrm{d}x = P\,\bar{u}(L) - \int_0^L \bar{u}\ddot{u}\rho A\,\mathrm{d}x, \tag{10.56}$$

where $\bar{u}(x)$ and $\bar{\varepsilon}(x) = \mathrm{d}u(x)/\mathrm{d}x$ are the virtual displacement and strain. Choosing

$$\bar{u}(x) = \sum_{m\,\text{odd}} \delta_m \sin \frac{\omega_m x}{c} \tag{10.57}$$

with arbitrary virtual displacement amplitudes δ_m, we have the equilibrium condition:

$$AE\sum_n\sum_m\int_0^L q_n\delta_m\frac{nm\pi^2}{4L^2}\cos\frac{n\pi x}{2L}\cos\frac{m\pi x}{2L}\,\mathrm{d}x$$
$$= P\sum_m \delta_m \sin\frac{m\pi}{2} - \rho A\sum_n\sum_m\int_0^L q_n\delta_m \sin\frac{n\pi x}{2L}\sin\frac{m\pi x}{2L}\,\mathrm{d}x. \tag{10.58}$$

The sums are on odd integers. Since

$$\int_0^L \sin\frac{n\pi x}{2L}\sin\frac{m\pi x}{2L}\,\mathrm{d}x = 0 \text{ for } n\neq m, \qquad \int_0^L\left(\sin\frac{n\pi x}{2L}\right)^2\mathrm{d}x = \frac{L}{2} \text{ for } n=m,$$
$$\int_0^L \cos\frac{n\pi x}{2L}\cos\frac{m\pi x}{2L}\,\mathrm{d}x = 0 \text{ for } n\neq m, \qquad \int_0^L\left(\cos\frac{n\pi x}{2L}\right)^2\mathrm{d}x = \frac{L}{2} \text{ for } n=m, \tag{10.59}$$

we find

$$\sum_m \delta_m\left[\frac{\rho AL}{2}\ddot{q}_m + \frac{AEm^2\pi^2}{8L}q_m - P\sin\frac{m\pi}{2}\right] = 0. \tag{10.60}$$

This relation must hold identically for all δ_m. Therefore,

$$\ddot{q}_m + \frac{m^2\pi^2c^2}{4L^2}q_m = \frac{2P}{\rho AL}\sin\frac{m\pi}{2}. \tag{10.61}$$

The solution for zero initial conditions is

$$q_m(t) = \frac{PL}{AE}\frac{8\sin(m\pi/2)}{m^2\pi^2}(1-\cos\omega_m t). \tag{10.62}$$

The displacement of the free end ($x = L$) is

$$u(L,t) = \frac{PL}{AE}\sum_{m\ \text{odd}}\frac{8}{m^2\pi^2}(1-\cos\omega_m t). \tag{10.63}$$

The maximum occurs when $t = 2L/c$ or $\omega_m t = m\pi$:

TABLE 10.2
Magnification Factor

Number of Modes	1	2	3	4
Magnification factor	1.621	1.801	1.866	1.899

$$\max u(L,t) = \frac{PL}{AE}\frac{16}{\pi^2}\sum_{n\ \text{odd}}\frac{1}{m^2}. \tag{10.64}$$

Since

$$\sum_{n\ \text{odd}}\frac{1}{m^2} = \frac{\pi^2}{8} \tag{10.65}$$

we find

$$\max u(L,t) = 2\frac{PL}{AE}. \tag{10.66}$$

That is, the dynamic displacement due to the step load is twice the static value, a dynamic magnification factor of 2. However, the series 10.65 converges slowly, so the magnification factor computed using any partial number of modes is much less than 2 as shown in Table 10.2.

We are able to find the exact solution to the continuum model of the rod (1D), but it is generally not possible to do so for 2D or 3D continuum models. The wave pattern is so complex that only numerical methods are feasible. In this case, the finite element model can provide a satisfactory approximation if properly interpreted.

10.5 EXAMPLE: SHORT BEAM

In Section 5.6.4, the short cantilever beam was analyzed as a plane stress problem using rectangular elements. The same layout (see Figure 10.10) will now be used for the determination of the lower frequencies and mode shapes.

For N unsupported nodes, there are $2N$ equations and therefore $2N$ natural frequencies. These frequencies approximate the lower natural frequencies of the body, which has of course infinitely many natural frequencies. Table 10.3 shows the results using the consistent mass matrix and the results using the lumped mass matrix together with the four-node rectangle (Section 15.17). The values are for ν is 1/3. The exact solution is unknown for this example.

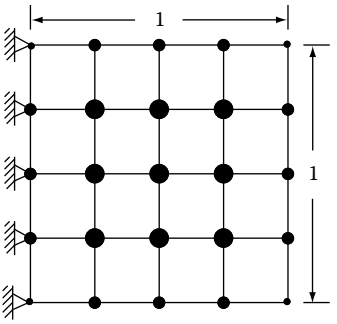


FIGURE 10.10 Lumped mass model.

TABLE 10.3
Short Beam

Number of Elements	Lumped Mass $\omega_1\sqrt{\rho a^2/E}$	Lumped Mass $\omega_2\sqrt{\rho a^2/E}$	Consistent Mass $\omega_1\sqrt{\rho a^2/E}$	Consistent Mass $\omega_2\sqrt{\rho a^2/E}$
1	0.59043	1.35962	0.77648	1.74559
16	0.66181	1.57645	0.67676	1.60033
100	0.65754	1.58141	0.65992	1.58531
256	0.65634	1.58154	0.65728	1.58311
10,000	0.65515	1.58123	0.65515	1.58129

10.6 DYNAMIC ANALYSIS WITH DAMPING

Damping was not included in the preceding analysis, but motions of real bodies are always damped. This damping is of little importance over a short time scale, but is essential for the analysis of the steady-state response. Damping arises from many dissipative processes accompanying the motion, the details of which are often not known. Damping is generally modeled by including a damping force proportional to velocities. The general equation 10.14 then becomes

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{V}\dot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{F}. \tag{10.67}$$

The damping matrix $\mathbf{V}$ is not generally determined by merging element matrices because the details of the damping mechanisms are not known. A typical assumption is*

$$\mathbf{V} = \alpha\mathbf{M} + \beta\mathbf{K} + \sum_i \beta_i \mathbf{K}_i \tag{10.68}$$

* The symbol α also occurs in the following formulation of numerical integration. To make it clear which alpha is meant, we may write alphad and betad for the damping coefficients.

where $\mathbf{K}_i$ is the part of the stiffness matrix due to material i. This is called Rayleigh damping or *proportional* damping. The coefficients are determined directly by experiments that measure the decay in amplitude of free vibrations. Proportional damping has some basis in theory, which we will now demonstrate.

10.6.1 Viscoelastic Damping

One damping mechanism is the internal dissipation of the material due to inelastic behavior. If the viscous loss is small and as there are no jumps in strain when inertia is included, the material may be adequately modeled by a two element Kelvin–Voigt model for both shearing and volume change:

$$\mathbf{s} = 2\mu\mathbf{e} + 2\eta\dot{\mathbf{e}}, \tag{10.69}$$

$$\frac{1}{3}\tau_{kk} = \kappa\varepsilon_{kk} + \zeta\dot{\varepsilon}_{kk}. \tag{10.70}$$

Since $s_{km} = \tau_{km} - \tau_{ii}\delta_{km}$ and $e_{km} = \varepsilon_{km} - \varepsilon_{ii}\delta_{km}$, Equations 10.69 and 10.70 are equivalent to the single constitutive equation

$$\tau_{km} = 2\mu\varepsilon_{km} + \lambda\varepsilon_{ii}\delta_{km} + 2\eta\dot{\varepsilon}_{km} + \lambda_{\mathrm{D}}\dot{\varepsilon}_{ii}\delta_{km}, \tag{10.71}$$

where

$$\lambda = \kappa - \frac{2}{3}\mu, \quad \lambda_{\mathrm{D}} = \zeta - \frac{2}{3}\eta. \tag{10.72}$$

There are four independent material constants: two elasticities and two viscosities. In matrix form, 10.71 is

$$\boldsymbol{\tau} = \mathbf{C}\boldsymbol{\varepsilon} + \mathbf{C}_{\mathrm{D}}\dot{\boldsymbol{\varepsilon}}, \tag{10.73}$$

where $\mathbf{C}$ and $\mathbf{C}_{\mathrm{D}}$ have the same form as in the elasticity problem if we define artificial Poisson ratios:

$$\nu = \frac{\lambda}{2\lambda + 2\mu}, \quad \nu_{\mathrm{D}} = \frac{\lambda_{\mathrm{D}}}{2\lambda_{\mathrm{D}} + 2\eta}. \tag{10.74}$$

The global equations of dynamical equilibrium are

$$\sum_n \int_{\mathcal{V}_n} \mathbf{A}^{\mathrm{T}}\boldsymbol{\tau}\mathrm{d}V = \mathbf{F} - \mathbf{M}\ddot{\mathbf{D}}, \tag{10.75}$$

where the summation denotes a merge of the element matrices, **F** is the global matrix of external nodal forces, and **M** is the mass matrix. Using 10.73 and $\boldsymbol{\varepsilon} = \mathbf{AD}$, the dynamical equilibrium equation 10.75 becomes

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{V}\dot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{F}, \tag{10.76}$$

where

$$\mathbf{K} = \sum_n \int_{\mathcal{V}_n} \mathbf{A}^{\mathrm{T}} \mathbf{C}\mathbf{A}\, \mathrm{d}V, \tag{10.77}$$

$$\mathbf{V} = \sum_n \int_{\mathcal{V}_n} \mathbf{A}^{\mathrm{T}} \mathbf{C}_{\mathrm{D}}\mathbf{A}\, \mathrm{d}V. \tag{10.78}$$

The matrix **K** is the familiar global stiffness matrix. The new matrix **V** is the matrix of damping coefficients, and we see from 10.76 that the effect of the viscoelastic material is to introduce Rayleigh damping with the coefficient matrix **V** into the basic equations of motion. In the special case when $\nu_{\mathrm{D}} = \nu$, the matrices **C** and $\mathbf{C}_{\mathrm{D}}$ are proportional. Let

$$\beta = \frac{\eta}{\mu}. \tag{10.79}$$

Then $\mathbf{C}_{\mathrm{D}} = \beta\mathbf{C}$ and

$$\mathbf{V} = \beta\mathbf{K}. \tag{10.80}$$

That is, the matrix of damping coefficients is a fraction of the stiffness matrix.

10.6.2 Viscous Body Force

If we imagine that there is at each point of the body an internal viscous force resisting the motion, it can be modeled by a body force proportional to the velocity of the particle:

$$b_i = -k\dot{u}_i. \tag{10.81}$$

The formula for nodal forces then gives the damping force

$$\mathbf{F}_{\mathrm{D}} = \sum_m \int_{\mathbf{V}_m} \mathbf{N}^{\mathrm{T}}\mathbf{b}\mathrm{d}V = -\sum_m \int_{\mathbf{V}_m} k\,\mathbf{N}^{\mathrm{T}}\dot{\mathbf{u}}\,\mathrm{d}V = -\left(\sum_m \int_{\mathbf{V}_m} k\,\mathbf{N}^{\mathrm{T}}\mathbf{N}\,\mathrm{d}V\right)\dot{\mathbf{D}} \tag{10.82}$$

after merging the element matrices. That is, the damping coefficient matrix is

$$\mathbf{V} = \sum_m \int_{\mathbf{V}_m} k\mathbf{N}^{\mathrm{T}}\mathbf{N}\,\mathrm{d}V. \tag{10.83}$$

If $\alpha = k/\rho$, we have

$$\mathbf{V} = \alpha\mathbf{M}, \tag{10.84}$$

where $\mathbf{M}$ is the consistent mass matrix.

10.6.3 Analysis of Damped Motion by Mode Superposition

The complete solution of the damped equations of motion can be constructed by superposition of the undamped modes:

$$\mathbf{D}(t) = \sum_n \mathbf{D}_n q_n(t). \tag{10.85}$$

Substituting in the equilibrium equations 10.67 and using the orthogonality conditions 10.23 and 10.24, in place of 10.26, we find

$$\ddot{q}_n + \sum_m d_{nm}\,\dot{q}_m + \omega_n^2 q_n = \frac{P_n}{M_n}, \tag{10.86}$$

where

$$d_{nm} = \frac{1}{M_n}\mathbf{D}_m^{\mathrm{T}}\mathbf{V}\mathbf{D}_n. \tag{10.87}$$

The equations are therefore coupled in general. However, for proportional damping $\mathbf{V} = \alpha\mathbf{M} + \beta\mathbf{K}$, using the orthogonality conditions, we have

$$d_{nm} = (\alpha + \beta\omega_n^2)\delta_{nm}. \tag{10.88}$$

In this case, the modal equations 10.86 separate:

$$\begin{aligned} &\ddot{q}_n + 2\zeta_n\omega_n\dot{q}_n + \omega_n^2 q_n = \frac{P_n}{M_n}, \\ &\zeta_n = \frac{\alpha + \beta\omega_n^2}{2\omega_n}. \end{aligned} \tag{10.89}$$

The general solution of each equation is that for damped motion of a single degree of freedom system. If the initial displacement and initial velocity are zero, for each n,

$$q(t) = \frac{1}{M\omega_{\mathrm{d}}} \int_0^t P(t-\tau) \mathrm{e}^{-\zeta\omega\tau} \sin(\omega_{\mathrm{d}}\tau)\, \mathrm{d}\tau, \tag{10.90}$$

$$\omega_{\mathrm{d}} = \omega\sqrt{1-\zeta^2}\ . \tag{10.91}$$

For a step load,

$$q(t) = \frac{P}{K}\left(1 - \mathrm{e}^{-\zeta\omega t}\left(\cos\omega_{\mathrm{d}}t + \zeta\frac{\omega}{\omega_{\mathrm{d}}}\sin\omega_{\mathrm{d}}t\right)\right) \tag{10.92}$$

for each n, which shows exponential decay due to damping.

10.7 NUMERICAL SOLUTION OF DIFFERENTIAL EQUATIONS

There are numerous techniques that have been used successfully for the numerical solution of differential equations. It appears, however, that best results for solid mechanics are obtained by using a method that is especially designed for such problems. We will consider first a scheme that is known under several names: constant average acceleration method, trapezoidal rule, or Newmark's method.

10.7.1 Constant Average Acceleration

For the purpose of calculating the value of a function $f(t)$ in the interval $a \le t \le b$, we assume a constant average acceleration:

$$\ddot{f}(t) = \frac{1}{2}(\ddot{f}_b + \ddot{f}_a). \tag{10.93}$$

The subscripts indicate the value at that time. Integration of 10.93 with respect to time gives

$$\dot{f}(t) = \dot{f}_a + \frac{(t-a)}{2}(\ddot{f}_b + \ddot{f}_a), \tag{10.94}$$

$$f(t) = f_a + (t-a)\dot{f}_a + \frac{(t-a)^2}{4}(\ddot{f}_b + \ddot{f}_a). \tag{10.95}$$

Now, put $t = b$ into 10.95, giving

$$\ddot{f}_b = \frac{4}{h^2}(f_b - f_a) - \frac{4}{h}\dot{f}_a - \ddot{f}_a. \tag{10.96}$$

Thus, expressing $\ddot{f}_b$ in terms of values of $\dot{f}_a$ and $\ddot{f}_a$ and the increment $f_b - f_a$ over the interval of time $h = b - a$.

Now we turn to the numerical solution of dynamical equations, step by step in time, using these approximations. Suppose the solution is known at time a. The fundamental equation applies at time b:

$$\mathbf{M}\ddot{\mathbf{D}}_b + \mathbf{K}\mathbf{D}_b = \mathbf{F}_b. \tag{10.97}$$

Equation 10.96 may now be used to eliminate the second derivatives to obtain

$$\bar{\mathbf{K}}\mathbf{D}_b = \bar{\mathbf{F}}_b, \tag{10.98}$$

where

$$\bar{\mathbf{K}} = \mathbf{K} + \frac{4}{h^2}\mathbf{M}, \tag{10.99}$$

$$\bar{\mathbf{F}}_b = \mathbf{F}_b + \frac{4}{h^2}\mathbf{M}\mathbf{D}_a + \frac{4}{h}\mathbf{M}\dot{\mathbf{D}}_a + \mathbf{M}\ddot{\mathbf{D}}_a, \tag{10.100}$$

and $h = b - a$ is the length of the time step. That is, the fundamental equation to be solved (10.98) is the same form as in the statical problem but with a modified stiffness matrix and an effective force matrix.

When $\mathbf{D}_a, \dot{\mathbf{D}}_a, \ddot{\mathbf{D}}_a$ are known, Equation 10.98 can be solved for $\mathbf{D}_b$. From 10.94 and 10.94, one then finds new values for velocity and acceleration:

$$\ddot{\mathbf{D}}_b = \frac{4}{h^2}(\mathbf{D}_b - \mathbf{D}_a) - \frac{4}{h}\dot{\mathbf{D}}_a - \ddot{\mathbf{D}}_a. \tag{10.101}$$

$$\dot{\mathbf{D}}_b = \dot{\mathbf{D}}_a + \frac{h}{2}(\ddot{\mathbf{D}}_a + \ddot{\mathbf{D}}_b). \tag{10.102}$$

The cycle is then repeated in order to march out a solution in finite time steps h.

10.7.2 General Newmark Method

Instead of using the average acceleration, the method can be improved by taking some value that better approximates the first integration over the interval. Let us consider first, in place of 10.94 and 10.95, the mathematical identities:

$$\dot{f}_b = \dot{f}_a + \int_a^b \ddot{f}(t)\,\mathrm{d}t, \tag{10.103}$$

$$f_b = f_a + h\dot{f}_a + \int_a^b (b-t)\ddot{f}(t)\,\mathrm{d}t. \tag{10.104}$$

Instead of constant acceleration, we can use the approximate integration formula

$$\int_a^b \ddot{f}(t)\,\mathrm{d}t \doteq W_1\ddot{f}_a + W_2\ddot{f}_b. \tag{10.105}$$

The weighting numbers W_i are determined by requiring that the integration formula yield the exact result in case $\ddot{f}$ is constant; this gives $W_1 + W_2 = h$, or in the usual notation

$$W_2 = \delta h, \quad W_1 = (1-\delta)h. \tag{10.106}$$

In the same way, we find

$$\int_a^b (b-t)\ddot{f}(t)\,\mathrm{d}t \doteq \left(\frac{1}{2}-\alpha\right)h^2\ddot{f}_a + \alpha h^2\ddot{f}_b\,. \tag{10.107}$$

Therefore, 10.103 and 10.104 give the fundamental approximations

$$\dot{f}_b = \delta h\ddot{f}_b + \dot{f}_a + (1-\delta)h\ddot{f}_a, \tag{10.108}$$

$$f_b = \alpha h^2\ddot{f}_b + f_a + h\dot{f}_a + \left(\frac{1}{2}-\alpha\right)h^2\ddot{f}_a\,, \tag{10.109}$$

in place of 10.96. The constant average acceleration formulas are the special case when $\delta = 1/2$ and $\alpha = 1/4$.*

10.7.3 General Methods

The Newmark scheme is only one way of marching the solution forward in time. A great many other systems have been worked out and analyzed. A general form of the relations is as follows:

$$\begin{aligned} f_b &= \alpha h^2 \ddot{f}_b + L_0(f_a, \dot{f}_a, \ddot{f}_a), \\ \dot{f}_b &= \delta h \ddot{f}_b + L_1(\dot{f}_a, \ddot{f}_a), \end{aligned} \tag{10.110}$$

where L_0 and L_1 are linear combinations of the indicated arguments and possibly of the values at other prior times, and α and δ are constants.

10.7.3.1 Implicit Methods in General

If α and δ are not zero, the system is called *implicit* because the value at time b is not explicitly determined by the state at time a. In this case, we can solve 10.110 for $\ddot{f}_b$ as was carried out above. Substitution of the result into 10.97 again yields 10.98 with

$$\begin{aligned} \bar{\mathbf{K}} &= \mathbf{K} + \frac{1}{\alpha h^2}\mathbf{M}, \\ \bar{\mathbf{F}}_b &= \mathbf{F}_b + \frac{1}{\alpha h^2}\mathbf{M}L_0(\mathbf{D}_a, \dot{\mathbf{D}}_a, \ddot{\mathbf{D}}_a). \end{aligned} \tag{10.111}$$

Solution of 10.98 gives f_b, and then 10.110 determines $\dot{f}_b$ and $\ddot{f}_b$.

10.7.3.2 Explicit Methods in General

If α and δ are zero, the scheme 10.110 is called *explicit* because the value of the function at time b is explicitly determined by the state at time a. In this case, we substitute 10.110 into 10.97 to obtain a formula for the second derivatives:

$$\ddot{\mathbf{D}}_b = \mathbf{M}^{-1}[\mathbf{F}_b - \mathbf{K}L_0(\mathbf{D}_a, \dot{\mathbf{D}}_a, \ddot{\mathbf{D}}_a)]. \tag{10.112}$$

Equations 10.110 with $\alpha = 0$ determine $\mathbf{D}_b$ and $\dot{\mathbf{D}}_b$.

$$\begin{aligned} \dot{\mathbf{D}}_b &= \mathbf{L}_1(\dot{\mathbf{D}}_a, \ddot{\mathbf{D}}_a), \\ \mathbf{D}_b &= \mathbf{L}_0(\mathbf{D}_a, \dot{\mathbf{D}}_a, \ddot{\mathbf{D}}_a). \end{aligned} \tag{10.113}$$

---

* ANSYS input $\delta = \frac{1}{2} + \gamma$, $\alpha = \frac{1}{4}(1+\lambda)^2$, default $\gamma = 0.005$.

Note that the stiffness matrix does not have to be inverted in the explicit method. Instead, the mass matrix is inverted. This is a trivial matter when **M** is diagonal as in the lumped mass model. We will not develop the solution by explicit methods in this book.

10.7.4 Stability Analysis of Newmark's Method

Let us consider the free vibration of the 1 degree of freedom system:

$$\ddot{f} + \omega^2 f = 0. \tag{10.114}$$

The exact solution is $f(t) = Ce^{i\omega t}$. At time $b, f_b = Ce^{i\omega b}$. At time $a, f_a = Ce^{i\omega a}$. Thus, $f_b = e^{i\omega h} f_a$, $h = b - a$. We seek a numerical solution of the same character but with possibly varying amplitude and somewhat different frequency: $f(t) = c(t)e^{i\bar{\omega}t}$. For this numerical solution, $f_b = \frac{c_b}{c_a} e^{i\bar{\omega}h} f_a$. That is,

$$f_b = \lambda f_a \tag{10.115}$$

where λ is a complex number. At time b, from 10.114, $\ddot{f}_b + \omega^2 f_b = 0$. Using 10.109, we find

$$f_b - f_a - h\dot{f}_a - \left(\frac{1}{2} - \alpha\right) h^2 \ddot{f}_a + \alpha h^2 \omega^2 f_b = 0. \tag{10.116}$$

Using 10.114 for $t = a$ to eliminate $\ddot{f}_a$, we have

$$(1 + \beta\alpha\omega^2) f_b = \left(1 - \left(\frac{1}{2} - \alpha\right) h^2 \omega^2\right) f_a + h\dot{f}_a. \tag{10.117}$$

Using 10.114 to eliminate the second derivatives in 10.108, we have

$$\delta h\omega^2 f_b + \dot{f}_b = -(1 - \delta) h\omega^2 f_a + \dot{f}_a. \tag{10.118}$$

Equations 10.117 and 10.118 have the form

$$\mathbf{A}\mathbf{u}_a = \mathbf{B}\mathbf{u}_b, \tag{10.119}$$

where

$$\mathbf{u} = \begin{bmatrix} f \\ \dot{f} \end{bmatrix}, \ \mathbf{A} = \begin{bmatrix} 1 - \left(\frac{1}{2} - \alpha\right) h^2\omega^2 & h \\ -(1 - \delta) h\omega^2 & 1 \end{bmatrix}, \ \mathbf{B} = \begin{bmatrix} (1 + \alpha h^2 \omega^2) & 0 \\ \delta h\omega^2 & 1 \end{bmatrix}. \tag{10.120}$$

We seek a solution of the form $\mathbf{u}_b = \lambda \mathbf{u}_a$, where λ is a complex number. That is,

$$(\mathbf{A} - \lambda \mathbf{B})\mathbf{u}_a = 0. \tag{10.121}$$

Setting the determinant of the coefficient matrix equal to zero, we find

$$\begin{aligned} &\lambda^2 - \left(2 - \left(\delta + \frac{1}{2}\right)k\right)\lambda + 1 - \left(\delta - \frac{1}{2}\right)k = 0, \\ &k = \frac{\omega^2 h^2}{1 + \alpha\omega^2 h^2}. \end{aligned} \tag{10.122}$$

Values for λ are complex if the discriminate is negative. This leads to

$$\left(\delta + \frac{1}{2}\right)^2 - 4\alpha < \frac{4}{\omega^2 h^2}. \tag{10.123}$$

Then

$$\lambda = \rho e^{i\theta}, \quad \rho = \left(1 - \left(\delta - \frac{1}{2}\right)k\right)^{1/2}. \tag{10.124}$$

The amplitude of the numerical solution will not grow, and therefore the solution procedure is stable, if $\rho \leq 1$ or $\delta \geq 1/2$. Using 10.123, the solution will be stable for all ω if

$$\delta \geq 1/2, \ \alpha \geq \frac{1}{4}\left(\delta + \frac{1}{2}\right)^2. \tag{10.125}$$

The constant average acceleration formula, with $\delta = 1/2$ and $\alpha = 1/4$, satisfies the criterion.

10.7.5 Convergence, Stability, and Error

The accuracy of various methods of direct time integration for particular differential equations has been studied in the mathematical theory of numerical analysis. The fundamental questions are convergence, stability, and error of the various schemes. Convergence implies that the numerical solution approaches the exact solution to the system 10.14 as the step interval h tends to zero. However, this is

not of much practical help. We must be able to obtain a sufficiently accurate solution for some finite value of h. Two difficulties can occur. The numerical solution may diverge rapidly from the true solution or oscillate wildly about it. In this case, the procedure is said to be unstable. It is generally found that an explicit scheme is stable if h does not exceed a limiting size, whereas an implicit scheme is stable for all h.

Second, the calculated result may preserve the essential features of the true solution and yet have an error at any particular time that is too large. The error has to be reduced by taking smaller time steps. We can see from the mode superposition solution that the applied load will tend to excite certain particular natural frequencies. A step load primarily excites the mode similar to the static solution. A variable load may excite higher frequencies. For acceptable error, the time step must be small compared to the period of the dominant mode shape that is excited by the load.

In the implicit method, the modified equilibrium equations 10.98 must be solved at each step. Implicit methods are typically stable for all values of h. The general Newmark method is unconditionally stable for $\delta \geq 1/2$, $\alpha \geq (\delta + 1/2)^2/4$. The interval h is therefore governed only by the need for computational accuracy. If we write*

$$\delta = \frac{1}{2} + \gamma, \quad \alpha = \frac{1}{4}(1+\gamma)^2, \tag{10.126}$$

then stability requires $\gamma \geq 0$. Good results are obtained for small values such as $\gamma = 0.005$, the default value for ANSYS. Note that the symbol α is not the same as the damping factor in 10.68.

The explicit schemes avoid solving the linear algebraic equations altogether. Typically, they are stable methods of numerical solution only if the magnitude of each time step is limited to some very small value, much less than the smallest period of vibration of the finite element model, although this is strongly affected by the damping mechanism. The advantage of an explicit method is that the equilibrium equations do not have to be solved. However, the requirement for a small time step may negate that advantage. Generally speaking, explicit methods are useful for rapidly varying loads (blast loading or crash analysis) where the time step has to be small anyway, or for systems with a small number of degrees of freedom.

10.7.6 Example: Numerical Integration for Axially Loaded Rod

One may solve Equation 10.44 using Newmark's method. The accuracy of the numerical integration depends on the size of the time steps. The numerical solution

* By means of Solution > Analyses Type > Sol'n Controls, the value of either γ or α and δ can be specified.

for $dt = 1$ is compared with the exact solution obtained by the mode superposition method in Figure 10.11.

The two curves will nearly coincide for $dt = 0.2$. For smaller time steps, convergence is to the exact solution of 10.44. The solution in the numerical procedure is an approximation to the finite element solution, not to the wave solution. We can only improve on the approximation to the wave equation by taking a larger number of elements.

Equation 10.44 was formulated using the lumped mass approximation. For only two elements, there is a big difference between the lumped mass model and the consistent mass model. The calculated natural frequencies are shown in Table 10.4 as $\omega\sqrt{\rho L^2/E}$ rad/s.

The numerical solution for the lumped mass model and the consistent mass model are compared in Figure 10.12, which shows the nondimensional displacement $D(t)/D_0$, $D_0 = PL/AE$, versus a nondimensional time defined by $\bar{t} = t\sqrt{E/\rho L^2}$ for $\Delta\bar{t} = 0.05$. Figure 10.13 shows the solution of the lumped mass model for different step sizes using the same parameters.

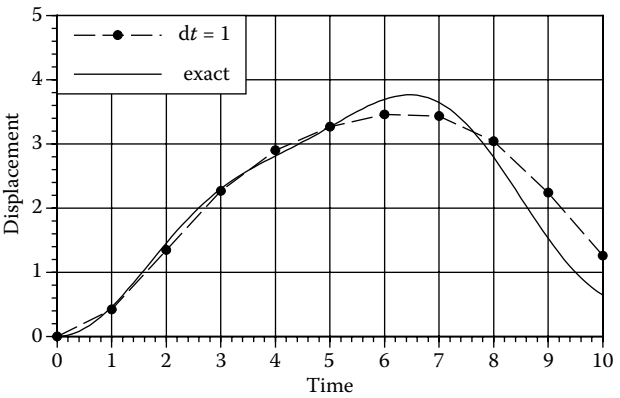


FIGURE 10.11 Comparison of mode superposition and numerical analysis.

TABLE 10.4
Two-Element Model

Frequency	Lumped	Consistent
1	1.5307	1.6114
2	3.6955	5.6293

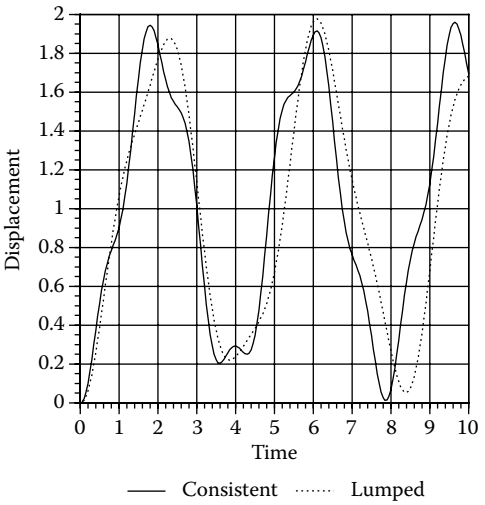


FIGURE 10.12 Lumped mass and consistent mass.

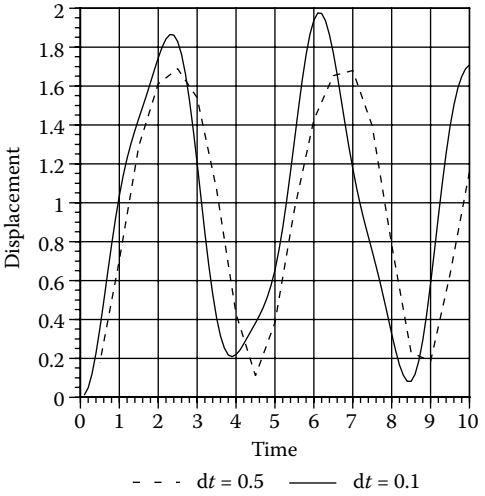


FIGURE 10.13 Lumped mass model.

10.8 EXAMPLE: ANALYSIS OF SHORT BEAM

Let us again consider the plane stress problem shown in Figure 10.14 and use the rectangular elements with lumped mass (Figure 10.10). We will use nondimensional values $a = 1$, $b = 1$, $p = 1$, $\rho = 1$, $E = 1$, $\nu = 1/3$. The element layout used was a 10×10 grid of 100 elements. The body is initially at rest and a step load p is applied. The calculated displacement of the corner without damping, using a time step $h = 0.1$, is plotted in Figure 10.15.

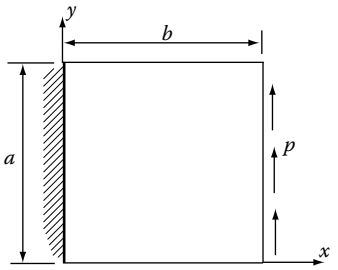


FIGURE 10.14 Short beam.

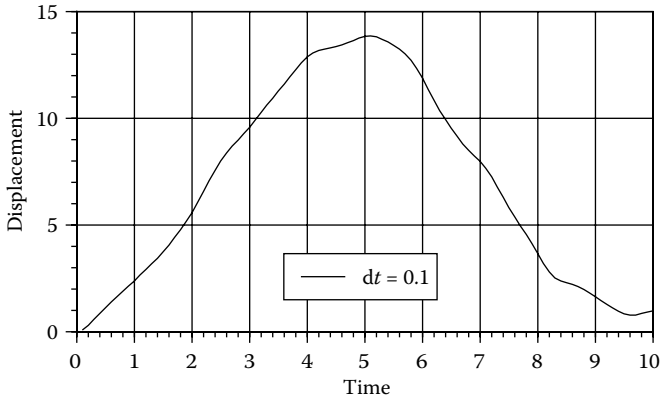


FIGURE 10.15 Numerical solution for the short beam.

We found in Section 5.6.4 that the calculated static displacement of the corner is 7.42. From Table 10.3, the first natural frequency of the discrete system is 0.66 rad/s and the period of vibration is $2\pi/\omega = 9.5$ s. For the step load, we expect a deflection that is about twice the static solution with a maximum at about 1/2 period. The calculated displacement of the corner versus time is plotted in Figure 10.15 and has the expected form. The exact solution is unknown.

An analysis including damping is shown in Figure 10.16 for viscous damping ($\alpha =$ 0.066) obtained using ANSYS (Newmark method, substep 0.1, 10×10 mesh). This is 5% of critical damping in mode 1. The result is shown in Figure 10.16 for the middle node on the free side. The magnification factor is reduced and the motion rapidly decays toward the static solution as expected.

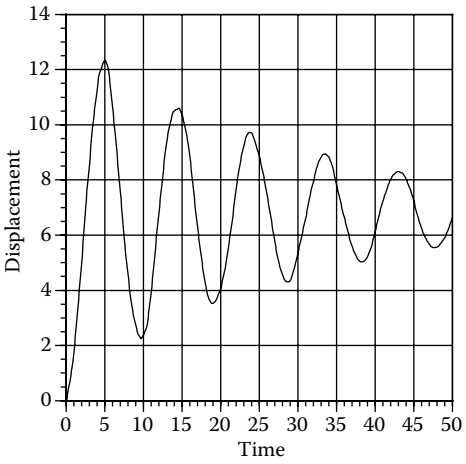


FIGURE 10.16 Damped motion.

10.9 PROBLEMS

1. Determine the mode shapes and frequencies for the rod shown in Figure 10.17 using two elements by solving the differential equation. Normalize the mode shapes so that $M_n = 1$.
2. Determine the mode shapes and frequencies for the rod shown in Figure 10.17 for two elements by using ANSYS and compare with the solution of problem 1. See Section 15.16. Note that ANSYS reports the frequencies in cps.
3. Consider a rod that is loaded at the end $x = 0$ and fixed at the end $x = L$ (see Figure 10.17). Divide the rod into two finite elements by a node at the center and use the lumped mass approach. For $A = 1$, $E = 1$, $\rho = 1$.
   (a) Solve by hand for the mode shapes and frequencies. Normalize the mode shapes so that the generalized mass is 1 for each mode.
   (b) Solve using ANSYS and compare results. See Section 15.16. Note that ANSYS reports frequencies in cps.

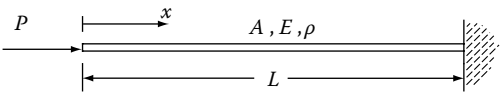


FIGURE 10.17 Axially loaded rod.

4. Consider a rod that is loaded at the end $x = 0$ and fixed at the end $x = L$ (see Figure 10.17). Divide the rod into two finite elements by a node at the center and use the lumped mass approach to derive the equations of motion.
   (a) Calculate the two natural frequencies of vibration.
   (b) Determine the nodal displacements for the case of a constant load suddenly applied to the end $x = 0$ at time $t = 0$.
   (c) Compare the finite element solution for the displacement at node 1 with the wave propagation analysis by plotting the displacement versus time.
5. Given the axially loaded rod shown in Figure 10.17 with $A = 1$, $L = 1$, $E = 100$, $\rho = 2$. The load is ramped up as shown in Figure 10.2 with $P_0 = 3$ and $t_1 = 1$. Use ANSYS to determine the displacement of the end for $0 \leq t \leq 2$. Use one element with lumped mass and numerical integration. Compare the maximum displacement with the exact solution (10.3.8)–(10.3.9).
6. Use ANSYS to determine the first two mode shapes and natural frequencies for the plane stress problem shown in Figure 10.18 for $a = b = 1$, $E = 1$, $\nu = 1/3$, $\rho = 1$. Use the four-node Plane 182 element and a 10×10 mesh. Determine the lowest two frequencies and view the mode shapes. See Section 15.17. (A printable list can be obtained via Utility Menu > List > Results > Load Step Summary.) Record the x component of displacements of nodes on the loaded edge for each mode for use in problem 7.

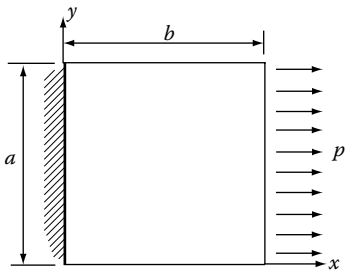


FIGURE 10.18 Plane stress extension.

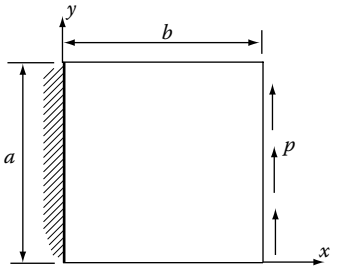


FIGURE 10.19 Plane stress bending.

7. Use the results of problem 6 to calculate the x displacement of the corner for step load $p = 10$ uniformly distributed over the edge $x = 1$ using only the first two modes.
8. Given the plane stress problem shown in Figure 10.19 with $a = b = 1$, $E = 1$, $\nu = 1/3$, $\rho = 1$, $p = 1$. Solve by ANSYS for a step load by the Newmark method $0 < t \leq 50$ using $dt = 0.1$ including damping with $\alpha = 0.066$. Use the four-node element without extra degree of freedom, and a 10×10 mesh. See Section 15.8. Submit a graph of UY at the center of the loaded side versus time (after adjusting colors to obtain black lines on a white background).
9. The finite element equations for dynamics with damping are $\mathbf{M}\ddot{\mathbf{D}} + \mathbf{V}\dot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{F}$. Determine the modified stiffness matrix $\bar{\mathbf{K}}$ and the modified force matrix $\bar{\mathbf{F}}$ when the Newmark method is used to solve these equations.

BIBLIOGRAPHY

Baruh, H., *Analytical Dynamics*. WCB/McGraw-Hill, New York, 1999.

Benaroya, H., and M. L. Nagurka, *Mechanical Vibration*, 3rd ed., CRC Press, Boca Raton, FL, 2010.

Bottega, W. J., *Engineering Vibrations*, CRC Press, Boca Raton, FL, 2006.

11 Linear Elastic Fracture Mechanics

11.1 FRACTURE CRITERION

If a crack has developed in a structure, the crack may propagate catastrophically when the stress intensity and the crack length combine to reach a critical situation. The basic formulation of the continuum theory of initiation of crack propagation will now be reviewed.*

Let us first consider an elastic sheet of thickness t with a plane crack of length $L = 2a$ through the sheet (Figure 11.1). The sheet is loaded in uniaxial tension S. After applying the load, there is a strain energy $\mathcal{U}$ stored in the body that depends on the magnitude of the load, the material properties, and the geometry of the body. In particular, $\mathcal{U}$ depends on the crack length L. If, at some load, the crack extends by amount dL, there is a release of the stored strain energy:

$$d\mathcal{U} = \frac{\partial \mathcal{U}}{\partial L} dL. \tag{11.1}$$

At the same time, energy of amount $d\mathbf{E}$ is expended to fracture the material and create a new free surface $dA = 2t dL$:

$$d\mathbf{E} = \mathbf{G} dA, \tag{11.2}$$

where $\mathbf{G}$ is the energy of crack growth per unit area. The fundamental assumption is that $\mathbf{G}$ is a material constant that has to be determined by materials testing. This is known as the Griffith hypothesis.

The energy balance for the crack extension requires that strain energy release rate must provide the energy needed for crack growth. Therefore, crack initiation can only occur if

$$\frac{\partial \mathcal{U}}{\partial L} = 2t\mathcal{G}. \tag{11.3}$$

The theory can be put in a more compact form if we define a so-called stress intensity factor K by

$$K = \sqrt{\frac{E}{t}\frac{\partial \mathcal{U}}{\partial L}} \tag{11.4}$$

* For a more complete treatment, see Dill: Chapter 6. Refer to Preface.

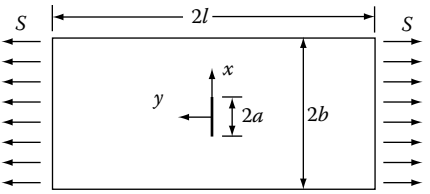


FIGURE 11.1 Cracked sheet in tension.

and let

$$K_c = \sqrt{2E\mathcal{G}}. \tag{11.5}$$

Then the condition for crack propagation becomes

$$K = K_c. \tag{11.6}$$

The critical stress intensity factor K_c is a material property to be determined by materials testing.

To determine K, we must perform a stress analysis of the body. The situation near the crack tip is certainly not modeled accurately by linear elasticity. The continuum model itself is probably not sufficient to describe the material behavior near the crack tip. However, a reasonably accurate prediction of strain energy release rate that is useful in engineering fracture mechanics results from this model.

Once the elasticity problem has been solved, the stored energy can be calculated. The strain energy per unit volume U of the material is such that

$$\tau_{ij} = \frac{\partial U}{\partial \varepsilon_{ij}} = 2\mu\varepsilon_{ij} + \lambda(\varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33})\delta_{ij}. \tag{11.7}$$

Therefore,

$$U = \frac{1}{2}\sum_k \sum_m \left(2\mu\varepsilon_{km}\varepsilon_{km} + \lambda\varepsilon_{kk}\varepsilon_{mm}\right) = \frac{1}{2}\sum_k \sum_m \left(\tau_{km}\varepsilon_{km}\right). \tag{11.8}$$

The total stored energy is

$$\mathcal{U} = \iiint U \, dV. \tag{11.9}$$

Integrating by parts and using the divergence theorem, neglecting the body force and the inertial force, we find that

$$\mathcal{U} = \frac{1}{2}\iint \mathbf{T} \cdot \mathbf{u} \, dA. \tag{11.10}$$

This equation will be used to calculate the stored energy.

11.1.1 Analysis of Sheet

The sheet is loaded in uniform tension in the y direction. The body is approximately in a state of plane stress ($\tau_{k3} = 0$). The complete solution is conveniently divided into two parts: the solution with no crack and the solution that removes the stress along the crack face. The solution for no crack is uniaxial tension:

$$\tau_{xx} = 0,\ \tau_{yy} = S,\ \tau_{xy} = 0. \tag{11.11}$$

This solution leaves a tensile stress $\tau_{yy} = S$ on each crack face. We therefore need to construct a solution for $\tau_{yy} = -S$ on each crack face (and no other loads) so that the combined solution will have zero tractions on the crack face. This second solution is known as the crack opening problem.

The exact solution of the crack opening problem for a body of finite extent is unknown, but if the crack is small, the solution for a body of infinite extend is a good approximation. This solution is known. The displacement of the crack surface is*

$$u_y = \frac{2S}{E}\sqrt{a^2 - x^2}\,. \tag{11.12}$$

The stress vector on the upper crack face is $\mathbf{T} = S\mathbf{j}$. The stored energy is therefore

$$\mathcal{U} = 4\left(\frac{1}{2}\int_0^a S\frac{2S}{E}\sqrt{a^2 - x^2}\,t\,\mathrm{d}x\right) = \frac{\pi L^2 S^2 t}{4E}, \tag{11.13}$$

where the crack length $L = 2a$. The total stored energy of the sheet is therefore

$$\mathcal{U} = \mathcal{U}_0 + \frac{\pi L^2 S^2 t}{4E} \tag{11.14}$$

where the first term is the energy for zero crack length. From this expression, we find

$$K = S\sqrt{\pi a}\,. \tag{11.15}$$

The solution for a body of finite dimensions has the form

$$K = \alpha S\sqrt{a\pi}\,, \tag{11.16}$$

where $\alpha > 1$ is a numerical factor that depends on the actual dimensions. The finite element method will be used to determine the factor α.

Crack growth initiates when S and crack length L reach critical levels such that

$$K = K_c. \tag{11.17}$$

* Dill, p. 243. Refer to Preface.

TABLE 11.1
Fracture Toughness

Material	Yield Stress	K_c (KSI$\sqrt{\text{in.}}$)
2024-T651 AL	66	22
7075-T651 AL	72	27
Ti–6AL–4V	124	84
4340 Steel (400°F)	235	50

Tests on fracture of cracked sheets show that the criterion (11.7) works fairly well except that the value of K_c depends on the sheet thickness. This might be expected since the stress state at the crack tip is less accurately described by plane stress for thick sheets and the crack extension is not well modeled by a plane surface. As sheet thickness increases, the value of K_c is found to approach a lower limit that is known as the fracture toughness in plane strain. Some typical values of the lower limit for K_c are shown in Table 11.1.[1]

11.1.2 Fracture Modes

There are three basic modes of crack opening that correspond to the remote state of stress.[2]

11.1.2.1 Mode I

The preceding analysis of the crack opening problem is applicable to any situation in which the stress state remote from the crack is a stress normal to the crack face: $\tau_{yy} = S$ and the other components are zero. This is known as a mode I crack. Let (r, θ) be the polar coordinates of a particle from the crack tip (Figure 11.2). Near $(r \ll a)$ the crack tip, the stress and displacement are, to first order, as follows:

$$\begin{aligned}
\tau_{xx} &= \frac{K_{\mathrm{I}}}{\sqrt{2\pi r}}\cos(\theta/2)\left(1-\sin(\theta/2)\sin(3\theta/2)\right),\\
\tau_{yy} &= \frac{K_{\mathrm{I}}}{\sqrt{2\pi r}}\cos(\theta/2)\left(1+\sin(\theta/2)\sin(3\theta/2)\right),\\
\tau_{xy} &= \frac{K_{\mathrm{I}}}{\sqrt{2\pi r}}\cos(\theta/2)\sin(\theta/2)\cos(3\theta/2),\\
u_x &= \frac{K_{\mathrm{I}}\sqrt{r}}{2G\sqrt{2\pi}}\cos(\theta/2)\left(\kappa-1+2\sin^2(\theta/2)\right),\\
u_y &= \frac{K_I\sqrt{r}}{2G\sqrt{2\pi}}\sin(\theta/2)\left(\kappa+1-2\cos^2(\theta/2)\right).
\end{aligned} \tag{11.18}$$

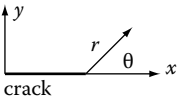


FIGURE 11.2 Local coordinates at crack tip.

where $K_{\mathrm{I}} = \lim(\sqrt{2\pi r}\ \tau_{yy}|_{\theta=0})_{r\to 0}$ and $\kappa = (3-\nu)/(1+\nu)$, $1+\kappa = 4/(1+\nu)$. This is the stress intensity factor for a mode I crack. For plane strain, $\kappa = 3-4\nu$ and $1+\kappa = 4(1-\nu)$.

11.1.2.2 Mode II

The case when the stress remote from the crack is $\tau_{xy} = S$ and the other components are zero (a simple shear in the direction of the crack) is called mode II. In this, case the stress near the crack tip is

$$\begin{aligned}
\tau_{xx} &= -\frac{K_{\mathrm{II}}}{\sqrt{2\pi r}}\sin(\theta/2)\Big(2+\cos(\theta/2)\cos(3\theta/2)\Big),\\
\tau_{yy} &= \frac{K_{\mathrm{II}}}{\sqrt{2\pi r}}\sin(\theta/2)\cos(\theta/2)\cos(3\theta/2),\\
\tau_{xy} &= \frac{K_{\mathrm{II}}}{\sqrt{2\pi r}}\cos(\theta/2)\Big(1-\sin(\theta/2)\sin(3\theta/2)\Big),\\
u_x &= \frac{K_{\mathrm{II}}\sqrt{r}}{2G\sqrt{2\pi}}\sin(\theta/2)\Big(\kappa+1+2\cos^2(\theta/2)\Big),\\
u_y &= -\frac{K_{\mathrm{II}}\sqrt{r}}{2G\sqrt{2\pi}}\cos(\theta/2)\Big(\kappa-1-2\sin^2(\theta/2)\Big),
\end{aligned} \tag{11.19}$$

where $K_{\mathrm{II}} = \lim(\sqrt{2\pi r}\ \tau_{xy}|_{\theta=0})_{r\to 0}$. This is the stress intensity factor for a mode II crack.

11.1.2.3 Mode III

The case when the stress remote from the crack is $\tau_{yz} = S$ and the other components are zero (a shear in the direction normal to the face of the sheet) is called mode III. In this case, the stress near the crack tip is

$$\begin{aligned}
\tau_{xz} &= -\frac{K_{\mathrm{III}}}{\sqrt{2\pi r}}\sin(\theta/2),\\
\tau_{yz} &= +\frac{K_{\mathrm{III}}}{\sqrt{2\pi r}}\cos(\theta/2),\\
u_z &= \frac{2K_{\mathrm{III}}\sqrt{r}}{G\sqrt{2\pi}}\sin(\theta/2),
\end{aligned} \tag{11.20}$$

where $K_{III} = \lim(\sqrt{2\pi r}\ \tau_{yz}|_{\theta=0})_{r\to 0}$.

When all three modes are present, if the crack propagates in the plane $y = 0$, the strain energy release rate for unit thickness is

$$\frac{d\mathcal{U}}{dL} = \frac{K_I^2}{E^*} + \frac{K_{II}^2}{E^*} + (1+\nu)\frac{K_{III}^2}{E} \tag{11.21}$$

where $E^* = E$ for plane stress and $E^* = E/(1-\nu^2)$ for plane strain.

11.2 DETERMINATION OF *K* BY FINITE ELEMENT ANALYSIS

For a complicated geometry or loading, the exact solution to the linear elasticity problem cannot be determined by direct means and we must turn to numerical methods. Once the stress analysis is completed, there are several methods that can be used to determine the stress intensity factor.

The oldest method is the direct calculation of the strain energy release rate. A stress analysis can be performed for various lengths L of a crack but the same external load. For each analysis, the stored energy is readily calculated. A curve of U versus L is plotted and the slope of this curve is the strain energy release rate. The stress intensity factor K is then determined by the fundamental definition 11.4. The accuracy of this method is limited since differentiation in order to determine the slope magnifies the error in the finite element calculation of displacements.

11.2.1 Crack Opening Displacement Method

Accuracy can be improved by using the finite element method to determine the crack opening displacement. This requires a detailed knowledge of displacements near the crack tip from the continuum mechanics analysis in advance of the finite element analysis. For example, in the case of the center crack in a thin sheet as is analyzed above, we have from 11.12 and 11.15 the following formula for displacements on the crack surface:

$$u_y = \frac{2K}{E\sqrt{\pi a}}\sqrt{a^2 - x^2}. \tag{11.22}$$

If r is the distance from the crack tip,

$$u_y = \frac{2K\sqrt{2r}}{E\sqrt{\pi}}\sqrt{1-\frac{r}{2a}} \approx \frac{4K\sqrt{r}}{E\sqrt{2\pi}}. \tag{11.23}$$

The displacement calculated at a node on the crack face near the crack tip can be used to determine u_y and the location of the node determines r. Equation 11.23 is then used to determine K. The accuracy of this calculation is strongly affected by the accuracy of the finite element model near the crack tip.

An alternative is the following procedure. From 11.18 with $\theta = \pi$, or 11.23 neglecting r/a, we have ($u_y \equiv v$) for plane stress

$$K = \frac{E\sqrt{2\pi}}{4}\frac{v}{\sqrt{r}}. \tag{11.24}$$

For plane strain

$$K = \frac{E\sqrt{2\pi}}{4(1-\nu^2)}\frac{v}{\sqrt{r}}. \tag{11.25}$$

The situation at the crack tip is a three-dimensional state somewhere between plane stress and plane strain, if the continuum model applies at all.

The displacement v normal to the crack is known from the finite element solution at the two node points on the crack surface and nearest to the crack tip. Let us designate them by points 1 and 2: $v = v_1$ at $r = r_1$ and $v = v_2$ at $r = r_2$, $r_2 > r_1$. Using a linear approximation for $v/\sqrt{r}$ in that region, we find the approximation

$$\frac{v}{\sqrt{r}} = \frac{1}{r_2 - r_1}\left(\frac{r_2 v_1}{\sqrt{r_1}} - \frac{r_1 v_2}{\sqrt{r_2}}\right). \tag{11.26}$$

This equation can be used to calculate $v/\sqrt{r}$ and therefore K from the finite element analysis.

As an example, the finite element analysis (Section 15.19) will be applied to the cracked sheet in tension (Figure 11.1), with $a = 2$, $b = 10$, and $l = 10$. Because of the symmetry about the x axis and about the y axis, we only need to analyze one-quarter of the sheet, $x > 0$ and $y > 0$. The boundary conditions along the edge $y = 0$ are $v = 0$ and $\tau_{xy} = 0$ for $x > a$, and both $\tau_{yy} = 0$ and $\tau_{xy} = 0$ on the crack surface $0 < x < a$. Along the edge $x = 0$, the boundary conditions are $u = 0$ and $\tau_{yx} = 0$. The nodal layout is shown in Figures 11.3 and 11.4. There are 283 nodes that are closely distributed around the singularity with larger elements in regions of lower stress. The six-node singular triangle is used at the crack tip and the eight-node quadrilateral elsewhere.

The nodes on the crack surface and the displacements are shown in Table 11.2.

The stress intensity factor using 11.26 with nodes 61 and 59 is $K = 2.648$. Using individual nodes and 11.23, the results for plane stress are shown in Table 11.3.

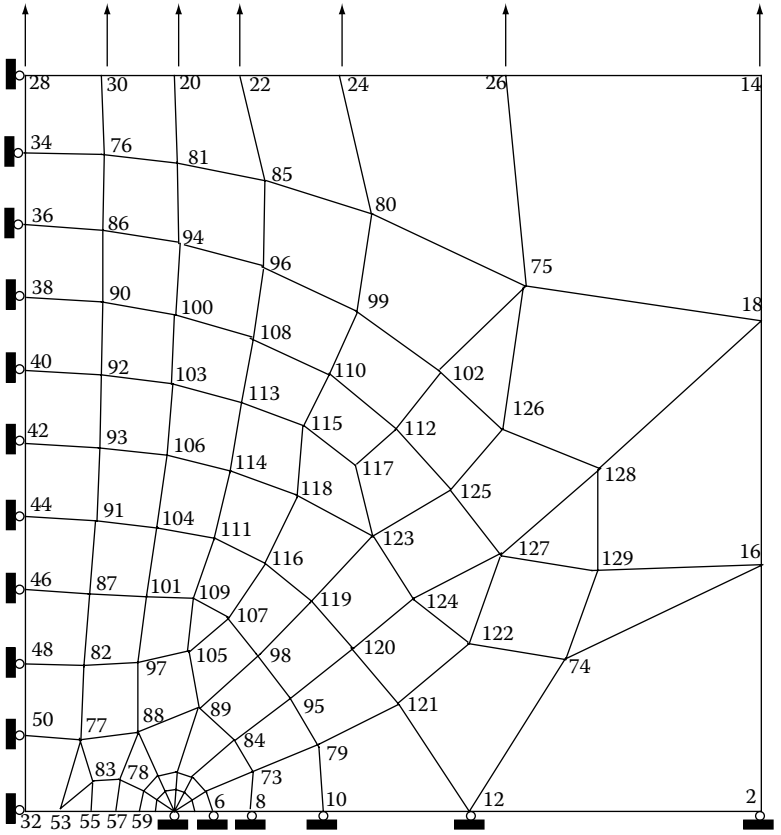


FIGURE 11.3 FEA nodes (corners only).

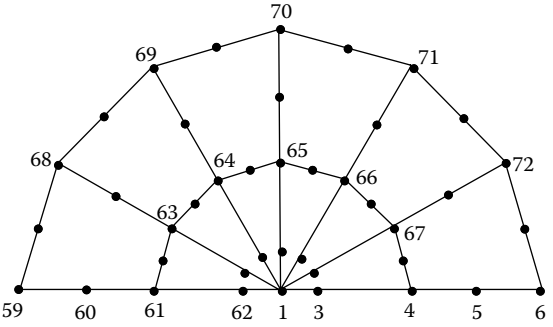


FIGURE 11.4 Nodes near the crack tip (node 1).

TABLE 11.2
Crack Opening

Node	*R*	*V*
62	0.0625	1.0502
61	0.2500	2.0442
60	0.3750	
59	0.5000	2.7943

TABLE 11.3
Stress Intensity Factor

Node	*K*
62	2.653
61	2.646
59	2.647

11.3 *J*-INTEGRAL FOR PLANE REGIONS

Let us consider the line integrals

$$I_m = \int_c f_{km} n_k \, \mathrm{d}s \tag{11.27}$$

around a closed curve c in the x–y plane, **n** is the outward normal, and

$$f_{km} = W\delta_{km} - \tau_{kj} \frac{\partial u_j}{\partial x_m} . \tag{11.28}$$

Since

$$\begin{aligned} \frac{\partial W}{\partial x_m} &= \frac{\partial W}{\partial \varepsilon_{rs}} \frac{\partial \varepsilon_{rs}}{\partial x_m} = \tau_{rs} \frac{\partial \varepsilon_{rs}}{\partial x_m} \\ &= \frac{1}{2} \tau_{rs} \left(\frac{\partial^2 u_r}{\partial x_m \partial x_s} + \frac{\partial^2 u_s}{\partial x_m \partial x_r} \right) = \tau_{kj} \frac{\partial^2 u_j}{\partial x_m \partial x_k} \end{aligned} \tag{11.29}$$

and

$$\frac{\partial \tau_{kj}}{\partial x_k} = 0. \tag{11.30}$$

For statical equilibrium without body force, we find that

$$\frac{\partial f_{km}}{\partial x_k} = \frac{\partial W}{\partial x_m} + \frac{\partial \tau_{kj}}{\partial x_k}\frac{\partial u_j}{\partial x_m} - \tau_{kj}\frac{\partial^2 u_j}{\partial x_m \partial x_k} = 0. \tag{11.31}$$

For a closed curve containing the region A, applying Gauss's theorem to 11.27, it follows that

$$I_m = \iint_A \frac{\partial f_{km}}{\partial x_k} \mathrm{d}A = 0. \tag{11.32}$$

That is, the integrals I_m are zero for any closed path c enclosing an area A in which the integrand possesses piecewise continuous first derivatives and does not contain singularities.

Let the closed curve be regarded as a path between two points (Figure 11.5). For the closed curve $c = c_1 + c_2$, we have

$$\begin{aligned} \oint_c f_{km} n_k \, \mathrm{d}s &= \left(\int_a^b f_{km} n_k \, \mathrm{d}s \right)_{c_1} + \left(\int_b^a f_{km} n_k \, \mathrm{d}s \right)_{c_2} \\ &= \left(\int_a^b f_{km} n_k \, \mathrm{d}s \right)_{c_1} - \left(\int_a^b f_{km} n_k \, \mathrm{d}s \right)_{c_2} = 0. \end{aligned} \tag{11.33}$$

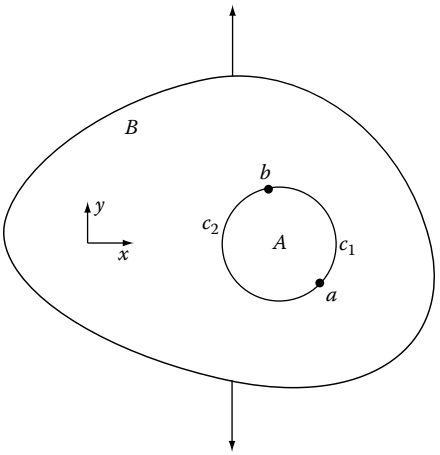


FIGURE 11.5 Two paths for integrals.

Consequently,

$$\left(\int_a^b f_{km} n_k \, \mathrm{d}s \right)_{c_1} = \left(\int_a^b f_{km} n_k \, \mathrm{d}s \right)_{c_2}. \tag{11.34}$$

That is, the integrals between two points are independent of the path between the points.

For an existing crack in plane stress or plane strain, we can choose the x axis parallel to the crack, and chose a path Γ starting and ending on the crack faces and enclosing a crack tip (Figure 11.6). The integral $J = I_1$ is called the J-integral:[3]

$$\begin{aligned} J &= \int_\Gamma f_k n_k \, \mathrm{d}s, \\ f_k &= W\delta_{k1} - \tau_{kj} \frac{\partial u_j}{\partial x_1}. \end{aligned} \tag{11.35}$$

We will show that J determines the energy release rate and therefore the stress intensity factors.

Recall that the integral is independent of the path. As a special case, we can choose the path Γ to be a circle of radius r about the crack tip and evaluate the J-integral as $r \to 0$. In the limit, only the singular terms in stress and strain formulas for the stress near the crack tip will contribute to the expression for J. After lengthy calculations using Mathematica, one finds

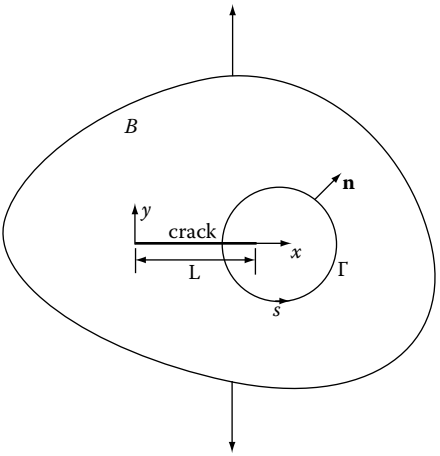


FIGURE 11.6 J-integral path.

$$J = \frac{K_{\mathrm{I}}^2}{E^*} + \frac{K_{\mathrm{II}}^2}{E^*} + (1+\nu)\frac{K_{\mathrm{III}}^2}{E}. \tag{11.36}$$

From 11.21, the J-integral is equal to the strain energy release rate:

$$J = \frac{\mathrm{d}\mathbf{U}}{\mathrm{d}L}. \tag{11.37}$$

Numerical calculation of the J-integral from 11.35 requires one to identify a path, calculate U, T_k, and $\partial u_k/\partial x$ along the path, and then integrate over the path. This is cumbersome in the context of the finite element analysis, and so a method has been devised to replace the line integral by integration over an area.[4]

Given a path Γ, construct a new path c surrounding it (Figure 11.7). Connect the two curves by paths along the crack surface, c^+ and c^-. For any smooth function $q(x,y)$,

$$\int_A f_k \frac{\partial q}{\partial x_k}\mathrm{d}a = \int_A \left(\frac{\partial}{\partial x_k}(qf_k) - q\frac{\partial f_k}{\partial x_k} \right)\mathrm{d}a = \oint_{\partial A} qf_k m_k \,\mathrm{d}s - \int_A q\frac{\partial f_k}{\partial x_k}\mathrm{d}a, \tag{11.38}$$

where **m** is the outward unit normal to A. First, we note that

$$\frac{\partial f_k}{\partial x_k} = 0. \tag{11.39}$$

Next, the arbitrary function $q(x,y)$ is given the values q = 1on Γ and $q = 0$ on c_1. On c^+ and c^-, $f_k\, m_k = 0$. Finally, on Γ, **m** = –**n**. Thus,

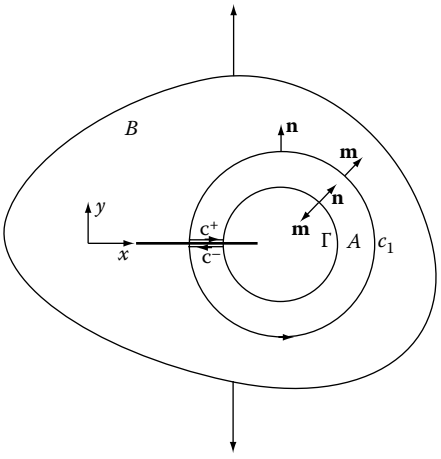


FIGURE 11.7 Area integral method.

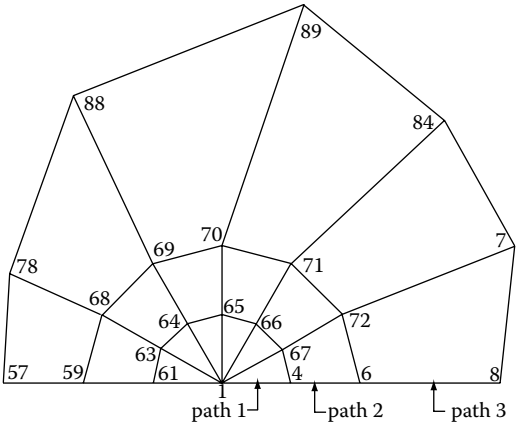


FIGURE 11.8 Element paths for *J*-integral evaluation.

$$\int_A f_k \frac{\partial q}{\partial x_k}\,\mathrm{d}a = -\oint_\Gamma q f_k n_k\,\mathrm{d}s = -J \tag{11.40}$$

or

$$J = -\int_A \left(W\delta_{k1} - \tau_{kj}\frac{\partial u_j}{\partial x_1} \right) \frac{\partial q}{\partial x_k}\,\mathrm{d}a. \tag{11.41}$$

The integral is evaluated over several element paths around the crack tip: J_1 uses the first ring of elements (Figure 11.8), J_2 uses the second ring, J_3 uses the third ring, etc. Because of the finite element approximation, the value of J is different for each path. The first path for J_1 is best discarded since it contains the singularity as a boundary. The true value for J is usually estimated as the average of the remaining paths.

$$J = \frac{1}{n-1}\left| J_2 + J_3 + \cdots + J_n \right| \tag{11.42}$$

11.4 PROBLEMS

1. Determine shape functions for a triangle with mid-side nodes from the eight-node quadrilateral by collapsing nodes 1, 4, and 8 together.
2. Given the shape functions for the four-node rectangle, obtain the shape functions for a right triangle by setting $X_1 = X_4$, $Y_1 = Y_4$, $U_1 = U_4$, and $V_1 = V_4$. Show that these shape functions agree with those for the three-node triangle if $\xi = 2\zeta_2 + 2\zeta_3 - 1$ and $\eta = (\zeta_3 - \zeta_2)/(\zeta_3 + \zeta_2)$.

3. For the singularity element (Section 5.9), given the shape functions 5.189 and the mapping 5.187, prove that $y = \frac{l}{4}\eta(1+\xi)^2$ for the triangular element with quarter-point nodes (Figure 5.28).
4. Use ANSYS to determine the stress intensity factor for a center crack ($a = 2$) in a 20 by 20 sheet assuming plane stress with $E = 1$, $\nu = 0.3$, by the crack opening method. See Section 15.19.
5. Determine the stress intensity factor for problem 4 using the *J*-integral method. See Section 15.20.

REFERENCES

1. Tapley, B. D. (ed.), *Eshback's Handbook of Engineering Fundamentals*, p. 478, Wiley, New York, 1990.
2. Liebowitz, H. (ed.) *Fracture*, Volume II, *Mathematical Fundamentals*, Academic Press, New York, 1968.
3. Rice, J. R., A path independent integral and the approximate analysis of strain concentration by notches and cracks, *J. Appl. Mech.*, 35, 379–386, 1968.
4. Shih, C. F., B. Moran, and T. Nakamura, Energy release rate along a three-dimensional crack front in a thermally stressed body, *Int. J. Fract.*, 30, 2, 79–102, 1986.

BIBLIOGRAPHY

Gdoutos, E. E., *Fracture Mechanics*, Springer, New York, 2005.

Kanninen, M. F., and C. H. Popelar, *Advanced Fracture Mechanics*, Oxford Univ. Press, London, 1985.

12 Plates and Shells

A plate is a thin sheet of material of thickness h. We choose a rectangular Cartesian coordinate system so that the middle surface of the plate lies in the x–y plane. The coordinate z is then normal to the middle surface. Loads acting on the plane of the middle surface cause a state of plane stress, and the plate remains a flat sheet. We now consider the deformation of thin sheets by loads that act normal to the plane of the middle surface and therefore produce bending of the plate. After deformation, the plate will no longer be flat. It will be slightly curved. The following analysis is limited to small deformations. The transverse displacement is limited to the order of magnitude of the plate thickness, and the radius of curvature at any point is large compared to the plate thickness.

12.1 GEOMETRY OF DEFORMATION

The components of displacement can be represented in a power series in the variable z. If the plate is thin enough, we can obtain a satisfactory approximation by retaining only the lower-order terms:

$$\begin{aligned} u(x,y,z) &\doteq u(x,y,0) + z\left.\frac{\partial u}{\partial z}\right|_{z=0}, \\ v(x,y,z) &\doteq v(x,y,0) + z\left.\frac{\partial v}{\partial z}\right|_{z=0}, \\ w(x,y,z) &\doteq w(x,y,0). \end{aligned} \tag{12.1}$$

That is,

$$\begin{aligned} u(x,y,z) &= \bar{u}(x,y) + z\alpha(x,y), \\ v(x,y,z) &= \bar{v}(x,y) + z\beta(x,y), \\ w(x,y,z) &= w(x,y). \end{aligned} \tag{12.2}$$

The components of the strain tensor are therefore given by

$$\begin{aligned} &\varepsilon_{xx} = \bar{\varepsilon}_{xx} + z\frac{\partial \alpha}{\partial x}, \quad \varepsilon_{yy} = \bar{\varepsilon}_{yy} + z\frac{\partial \beta}{\partial y}, \quad \gamma_{xy} = \bar{\gamma}_{xy} + z\left(\frac{\partial \alpha}{\partial y} + \frac{\partial \beta}{\partial x}\right), \\ &\gamma_{yz} = \frac{\partial w}{\partial y} + \beta, \quad \gamma_{zx} = \frac{\partial w}{\partial x} + \alpha. \end{aligned} \tag{12.3}$$

The in-plane strains for $z = 0$ are

$$\bar{\varepsilon}_{xx} = \frac{\partial \bar{u}}{\partial x}, \quad \bar{\varepsilon}_{yy} = \frac{\partial \bar{v}}{\partial y}, \quad \bar{\gamma}_{xy} = \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x}. \tag{12.4}$$

The approximate expression 12.2 cannot be used to calculate ε_{zz}.

12.2 EQUATIONS OF EQUILIBRIUM

We will first calculate the stress resultants on cross sections of the plate along the coordinate lines.

The sign convention is shown in Figure 12.1. On the cross section normal to the x axis, the resultant forces and resultant moments about the axes per unit edge length are

$$N_x = \int_{-h/2}^{+h/2} \tau_{xx}\, dz, \quad N_{xy} = \int_{-h/2}^{+h/2} \tau_{xy}\, dz. \tag{12.5}$$

$$Q_x = \int_{-h/2}^{+h/2} \tau_{xz}\, dz, \quad M_x = \int_{-h/2}^{+h/2} z\tau_{xx}\, dz, \quad M_{xy} = \int_{-h/2}^{+h/2} z\tau_{xy}\, dz. \tag{12.6}$$

On the cross section normal to the y axis the resultant forces and resultant moments about the axes per unit edge length are

$$N_y = \int_{-h/2}^{+h/2} \tau_{yy}\, dz, \quad N_{xy} = \int_{-h/2}^{+h/2} \tau_{xy}\, dz. \tag{12.7}$$

$$Q_y = \int_{-h/2}^{+h/2} \tau_{yz}\, dz, \quad M_y = \int_{-h/2}^{+h/2} z\tau_{yy}\, dz, \quad M_{yx} = \int_{-h/2}^{+h/2} z\tau_{xy}\, dz. \tag{12.8}$$

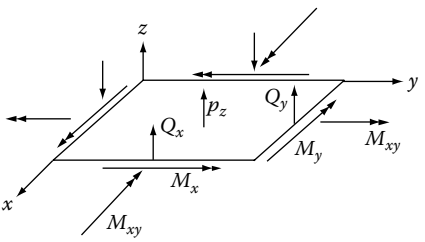


FIGURE 12.1 Plate sign convention.

The condition of equilibrium of an infinitesimal segment dx by dy of the plate evidently requires the stress resultants to satisfy the following differential equations. From a summation of forces in the x and y directions, we find

$$\begin{aligned} \frac{\partial N_x}{\partial x} + \frac{\partial N_{xy}}{\partial y} + p_x &= 0, \\ \frac{\partial N_y}{\partial y} + \frac{\partial N_{xy}}{\partial x} + p_y &= 0. \end{aligned} \tag{12.9}$$

From a summation of forces in the z direction and a summation of moments about the x and y axes, we find

$$\begin{aligned} \frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} + p_z &= 0, \\ \frac{\partial M_x}{\partial x} + \frac{\partial M_{xy}}{\partial y} - Q_x &= 0, \\ \frac{\partial M_y}{\partial y} + \frac{\partial M_{xy}}{\partial x} - Q_y &= 0. \end{aligned} \tag{12.10}$$

The applied loads per unit of plate area are p_x, p_y, and p_z. The set 12.9 is the integrated equations of the theory of plane stress for in-plane loading of the sheet. The system 12.10 is associated with the bending of the plate.

12.3 CONSTITUTIVE RELATIONS FOR AN ELASTIC MATERIAL

Since the plate is thin, the stress component τ_{zz} is of the order of magnitude of the loading on the exterior surfaces of the sheet and much smaller than the primary stresses in bending τ_{xx} and τ_{yy}. It can be neglected in the stress–strain relations, so that

$$\varepsilon_{xx} \doteq \frac{1}{E}(\tau_{xx} - \nu\tau_{yy}), \quad \varepsilon_{yy} \doteq \frac{1}{E}(\tau_{yy} - \nu\tau_{xx}). \tag{12.11}$$

Therefore,

$$\begin{aligned} &\tau_{xx} = \frac{E}{1-\nu^2}(\varepsilon_{xx} + \nu\varepsilon_{yy}), \quad \tau_{yy} = \frac{E}{1-\nu^2}(\varepsilon_{yy} + \nu\varepsilon_{xx}), \quad \tau_{xy} = \frac{E}{2(1+\nu)}\gamma_{xy}, \\ &\tau_{yz} = \frac{E}{2(1+\nu)}\gamma_{yz}, \quad \tau_{zx} = \frac{E}{2(1+\nu)}\gamma_{zx}. \end{aligned} \tag{12.12}$$

From 12.3, 12.5, 12.7, and 12.12, the in-plane stress resultants have the constitutive relations:

$$N_x = \frac{Eh}{(1-\nu^2)}\left(\bar{\varepsilon}_{xx} + \nu\bar{\varepsilon}_{yy}\right),$$
$$N_y = \frac{Eh}{(1-\nu^2)}\left(\bar{\varepsilon}_{yy} + \nu\bar{\varepsilon}_{xx}\right), \tag{12.13}$$
$$N_{xy} = \frac{Eh}{2(1+\nu)}\bar{\gamma}_{xy}.$$

From 12.1, 12.6, 12.8, and 12.12, the stress resultants in bending are related to the deformations by the following constitutive relations:

$$M_x = \frac{Eh^3}{12(1-\nu)^2}\left(\frac{\partial\alpha}{\partial x} + \nu\frac{\partial\beta}{\partial y}\right),$$
$$M_y = \frac{Eh^3}{12(1-\nu)^2}\left(\frac{\partial\beta}{\partial y} + \nu\frac{\partial\alpha}{\partial x}\right), \tag{12.14}$$
$$M_{xy} = \frac{Gh^3}{12}\left(\frac{\partial\alpha}{\partial y} + \frac{\partial\beta}{\partial x}\right),$$

$$Q_x = Gh\gamma_{xz}, \quad Q_y = Gh\gamma_{yz}. \tag{12.15}$$

The constitutive relations 12.15 for the shear resultants can be improved as follows. The strain energy of the transverse shears is

$$\begin{aligned}\mathcal{U} &= \frac{1}{2}\int\int(\tau_{xz}\gamma_{xz} + \tau_{yz}\gamma_{yz})\,\mathrm{d}z\,\mathrm{d}A \\ &= \frac{1}{2}\int\int\left(\frac{1}{G}\tau_{xz}^2 + \frac{1}{G}\tau_{yz}^2\right)\mathrm{d}z\,\mathrm{d}A.\end{aligned} \tag{12.16}$$

If there are no surface tractions, the shear stress is zero for $z = \pm h/2$ and symmetric about the middle surface. To first approximation, we presume it to be parabolically distributed as for the bending of beams:

$$\tau_{xz} = \frac{3}{2}\bar{\tau}_{xz}\frac{4}{h^2}\left(\frac{h^2}{4} - z^2\right),$$
$$\tau_{yz} = \frac{3}{2}\bar{\tau}_{yz}\frac{4}{h^2}\left(\frac{h^2}{4} - z^2\right), \tag{12.17}$$

where $\bar{\tau}_{xz} = Q_x/h$ and $\bar{\tau}_{yz} = Q_y/h$ are the average shear stresses. Then, substituting these expressions into 12.16, we find

$$\mathcal{U} = \frac{1}{2}\int\left(\frac{6h}{5G}\overline{\tau}_{xz}^2 + \frac{6h}{5G}\overline{\tau}_{yz}^2\right)\mathrm{d}A. \tag{12.18}$$

We now require that the strain energy of the plate be accurately given by expression 12.16 but using the average shear stress and the strains at $z = 0$:

$$\begin{aligned}\mathcal{U} &= \frac{1}{2}\int\int\left(\overline{\tau}_{xz}\gamma_{xz} + \overline{\tau}_{yz}\gamma_{yz}\right)\mathrm{d}z\,\mathrm{d}A \\ &= \frac{1}{2}\int\left(h\overline{\tau}_{xz}\gamma_{xz} + h\overline{\tau}_{yz}\gamma_{yz}\right)\mathrm{d}A.\end{aligned} \tag{12.19}$$

This formula will agree with 12.18 if

$$\overline{\tau}_{xz} = kG\gamma_{xz}, \quad \overline{\tau}_{yz} = kG\gamma_{yz}, \quad k = \frac{5}{6}. \tag{12.20}$$

Therefore, the constitutive relations 12.15 become

$$Q_x = kGh\gamma_{xz}, \quad Q_x = kGh\gamma_{xz}, \tag{12.21}$$

where k is a shear correction factor, approximately 5/6. These relations are used in place of 12.15.

12.4 VIRTUAL WORK

The complete set of equations for the bending part consists of the geometric relations from 12.3 combined with the constitutive relations 12.14 and 12.21,

$$\begin{aligned}Q_x &= kGh\left(\frac{\partial w}{\partial x} + \alpha\right), \\ Q_y &= kGh\left(\frac{\partial w}{\partial y} + \beta\right), \\ M_x &= \frac{Eh^3}{12(1-\nu)^2}\left(\frac{\partial \alpha}{\partial x} + \nu\frac{\partial \beta}{\partial y}\right), \\ M_y &= \frac{Eh^3}{12(1-\nu)^2}\left(\frac{\partial \beta}{\partial y} + \nu\frac{\partial \alpha}{\partial x}\right), \\ M_{xy} &= \frac{Eh^3}{24(1+\nu)}\left(\frac{\partial \alpha}{\partial y} + \frac{\partial \beta}{\partial x}\right),\end{aligned} \tag{12.22}$$

and the equilibrium equations 12.10,

$$\begin{aligned}\frac{\partial Q_x}{\partial x}+\frac{\partial Q_y}{\partial y}+p_z&=0,\\ \frac{\partial M_x}{\partial x}+\frac{\partial M_{xy}}{\partial y}-Q_x&=0,\\ \frac{\partial M_y}{\partial y}+\frac{\partial M_{xy}}{\partial x}-Q_y&=0,\end{aligned}\tag{12.23}$$

where we drop the bar over the middle surface displacement w so that the bar can be used for virtual displacements.

The theorem of virtual work can be obtained by multiplying each equilibrium equation by a virtual displacement $\bar{w}$ and virtual rotations $\bar{\alpha}$ and $\bar{\beta}$ and integrating over the area of the plate:

$$\int\left\{\left(\frac{\partial Q_x}{\partial x}+\frac{\partial Q_y}{\partial y}+p_z\right)\bar{w}+\left(\frac{\partial M_x}{\partial x}+\frac{\partial M_{xy}}{\partial y}-Q_x\right)\bar{\alpha}+\left(\frac{\partial M_y}{\partial y}+\frac{\partial M_{xy}}{\partial x}-Q_y\right)\bar{\beta}\right\}\mathrm{d}A=0.\tag{12.24}$$

Integrating by parts gives

$$\begin{aligned}&\int_A\left\{Q_x\left(\frac{\partial\bar{w}}{\partial x}+\bar{\alpha}\right)+Q_y\left(\frac{\partial\bar{w}}{\partial y}+\bar{\beta}\right)+M_x\frac{\partial\bar{\alpha}}{\partial x}+M_{xy}\left(\frac{\partial\bar{\alpha}}{\partial y}+\frac{\partial\bar{\beta}}{\partial x}\right)+M_y\frac{\partial\bar{\beta}}{\partial y}\right\}\mathrm{d}A\\ &=\int_A p_z\bar{w}\,\mathrm{d}A+\int_C\left\{\left(Q_xn_x+Q_yn_y\right)\bar{w}+(M_xn_x+M_{xy}n_y)\bar{\alpha}+(M_{xy}n_x+M_yn_y)\bar{\beta}\right\}\mathrm{d}s.\end{aligned}\tag{12.25}$$

The left-hand side is the internal virtual work of the stress resultants and the right-hand side is the external virtual work of the edge loads.

The virtual work formula shows that α is a rotation in the direction of M_x and β is a rotation in the direction of M_y. The positive directions are shown in Figure 12.1. Let θ_x = ROTX and θ_y = ROTY denote rotations about the respective axes with positive directions determined by the right-hand rule, then

$$\begin{aligned}\theta_x&=-\beta,\\ \theta_y&=+\alpha.\end{aligned}\tag{12.26}$$

For matrix notation, we define

$$\mathbf{Q} = \begin{bmatrix} Q_x \\ Q_y \end{bmatrix}, \quad \boldsymbol{\gamma} = \begin{bmatrix} \dfrac{\partial w}{\partial x} + \alpha \\ \dfrac{\partial w}{\partial y} + \beta \end{bmatrix}, \tag{12.27}$$

$$\mathbf{M} = \begin{bmatrix} M_x \\ M_y \\ M_{xy} \end{bmatrix}, \quad \boldsymbol{\kappa} = \begin{bmatrix} \dfrac{\partial \alpha}{\partial x} \\ \dfrac{\partial \beta}{\partial y} \\ \dfrac{\partial \alpha}{\partial y} + \dfrac{\partial \beta}{\partial x} \end{bmatrix}. \tag{12.28}$$

The constitutive relations are then

$$\mathbf{Q} = \mathbf{C}_\mathrm{s}\,\gamma \tag{12.29}$$

$$\mathbf{M} = \mathbf{C}_\mathrm{b}\kappa, \tag{12.30}$$

where

$$\mathbf{C}_\mathrm{b} = \frac{Eh^3}{12(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix}, \quad \mathbf{C}_\mathrm{s} = \frac{Ehk}{2(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}. \tag{12.31}$$

The internal virtual work can then be written as

$$\int_A \left(\bar{\boldsymbol{\gamma}}^\mathrm{T} \mathbf{Q} + \bar{\boldsymbol{\kappa}}^\mathrm{T} \mathbf{M} \right) \mathrm{d}A = \int_A \left(\bar{\boldsymbol{\gamma}}^\mathrm{T} \mathbf{C}_\mathrm{s} \boldsymbol{\gamma} + \bar{\boldsymbol{\kappa}}^\mathrm{T} \mathbf{C}_\mathrm{b} \boldsymbol{\kappa} \right) \mathrm{d}A \tag{12.32}$$

where the bar indicates the virtual strains and curvatures.

The system of Equations 12.22 and 12.23 is of sixth order so the solution can satisfy three boundary conditions on each edge. The appropriate combination can be seen from the virtual work formula, 12.25:

$$\left.\begin{array}{l} w \text{ or } Q_x, \\ \alpha \text{ or } M_x, \\ \beta \text{ or } M_{xy}, \end{array}\right\} \text{given on the edge } x = \text{constant}, \tag{12.33}$$

and

$$\left.\begin{array}{l} w \text{ or } Q_y, \\ \alpha \text{ or } M_{xy}, \\ \beta \text{ or } M_y, \end{array}\right\} \text{given on the edge } y = \text{constant.} \tag{12.34}$$

An edge (say x = constant) for which $w = 0$ and $M_x = 0$ is said to be "simply supported." There are therefore two kinds of simply supported edges: (1) $\beta = 0$ when the edge is supported by a rigid membrane that can provide the necessary twisting moments and (2) $M_{xy} = 0$ when the support provides no restraint for rotation about the normal.

12.5 FINITE ELEMENT RELATIONS FOR BENDING

For the reduction to finite element formulation, we need shape functions for the displacement w and the rotations α and β. However, direct implementation with lower-order shape functions has been found to introduce excessive stiffening in shear, known as shear locking, and the convergence for thin plates is slow or nonexistent. Numerous alternatives have been developed. We will present a four-node rectangular element that has been shown to work well.[1–3]

The degrees of freedom are the values of w, α, and β at the four nodes:

$$\mathbf{D}^{\mathrm{T}} = \begin{bmatrix} w_1 & w_2 & w_3 & w_4 & \alpha_1 & \alpha_2 & \alpha_3 & \alpha_4 & \beta_1 & \beta_2 & \beta_3 & \beta_4 \end{bmatrix} \tag{12.35}$$

where w_i, α_i, and β_i are the nodal values of the parameters. The serendipity functions are used for interpolation of the functions:

$$\alpha = \sum_i N_i \alpha_i, \quad \beta = \sum_i N_i \beta_i, \quad w = \sum_i N_i w_i. \tag{12.36}$$

The shape functions are given by 5.10 and their derivatives by 5.13. From 12.28,

$$\kappa = \mathbf{A}_{\mathrm{b}} \mathbf{D}, \tag{12.37}$$

where $\mathbf{A}_{\mathrm{b}}$ is given by 5.20 augmented by zeros for the w_i degrees of freedom:

$$\mathbf{A}_{\mathrm{b}} = \begin{array}{c} \begin{array}{ccc} \mathbf{w} & \boldsymbol{\alpha} & \boldsymbol{\beta} \end{array} \\ \begin{bmatrix} 0 & \mathbf{A}_1 & \mathbf{0} \\ 0 & \mathbf{0} & \mathbf{A}_2 \\ 0 & \mathbf{A}_2 & \mathbf{A}_1 \end{bmatrix} \end{array}. \tag{12.38}$$

The bending part of the virtual work 12.32 is

$$\int_A \left(\bar{\boldsymbol{\kappa}}^{\mathrm{T}}\mathbf{C}_{\mathrm{b}}\boldsymbol{\kappa}\right)\mathrm{d}a = \bar{\mathbf{D}}^{\mathrm{T}}\mathbf{k}_{\mathrm{b}}\mathbf{D},$$
$$\mathbf{k}_{\mathrm{b}} = \int_A \mathbf{A}_{\mathrm{b}}^{\mathrm{T}}\mathbf{C}_{\mathrm{b}}\mathbf{A}_{\mathrm{b}}\,\mathrm{d}a. \tag{12.39}$$

Since $\mathbf{C}_{\mathrm{b}}$ is just $h^3/12$ times the elasticity matrix for plane stress, the stiffness matrix $\mathbf{k}_{\mathrm{b}}$ is just $h^3/12$ times the stiffness matrix for the four node rectangle, augmented by zeros for the w terms:

$$\mathbf{k}_{\mathrm{b}} = \begin{bmatrix} \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{k}_{\mathrm{b}}^{22} & \mathbf{k}_{\mathrm{b}}^{23} \\ \mathbf{0} & \mathbf{k}_{\mathrm{b}}^{32} & \mathbf{k}_{\mathrm{b}}^{33} \end{bmatrix}. \tag{12.40}$$

For the square element, with $\nu = 1/3$, from 5.27:

$$\begin{bmatrix} \mathbf{k}_b^{22} & \mathbf{k}_b^{23} \\ \mathbf{k}_b^{32} & \mathbf{k}_b^{33} \end{bmatrix} = \frac{D(1-\nu)}{48} \begin{bmatrix} 32 & -20 & -16 & 4 & 12 & 0 & -12 & 0 \\ -20 & 32 & 4 & -16 & 0 & -12 & 0 & 12 \\ -16 & 4 & 32 & -20 & -12 & 0 & 12 & 0 \\ 4 & -16 & -20 & 32 & 0 & 12 & \mathbf{0} & -12 \\ 12 & 0 & -12 & 0 & 32 & 4 & -16 & -20 \\ 0 & -12 & 0 & 12 & 4 & 32 & -20 & -16 \\ -12 & 0 & 12 & 0 & -16 & -20 & 32 & 4 \\ 0 & 12 & 0 & -12 & -20 & -16 & 4 & 32 \end{bmatrix}. \tag{12.41}$$

To avoid excessive influence of the shear term, full integration of the shear term in the virtual work formula is not used. Instead, further approximation of the shear strains is made as follows. From 12.4 and 12.36, at points A, B, C, and D, in Figure 12.2,

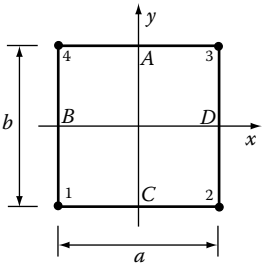


FIGURE 12.2 Four-node plate element.

$$\begin{aligned}
\gamma_{xz}^{A} &= \frac{w_3 - w_4}{a} + \frac{\alpha_3 + \alpha_4}{2}, \\
\gamma_{xz}^{C} &= \frac{w_2 - w_1}{a} + \frac{\alpha_2 + \alpha_1}{2}, \\
\gamma_{yz}^{D} &= \frac{w_3 - w_2}{b} + \frac{\beta_3 + \beta_2}{2}, \\
\gamma_{yz}^{B} &= \frac{w_4 - w_1}{b} + \frac{\beta_4 + \beta_1}{2}.
\end{aligned} \tag{12.42}$$

The shear for the element is now interpolated linearly between the two opposite points:

$$\begin{aligned}
\gamma_{xz} &= \frac{1}{2}(1+\eta)\gamma_{xz}^{A} + \frac{1}{2}(1-\eta)\gamma_{xz}^{C}, \\
\gamma_{yz} &= \frac{1}{2}(1+\xi)\gamma_{yz}^{D} + \frac{1}{2}(1-\xi)\gamma_{yz}^{B}.
\end{aligned} \tag{12.43}$$

Combining these formulas, we have the representation for the shears:

$$\boldsymbol{\gamma} = \mathbf{A}_s\mathbf{D}, \tag{12.44}$$

where

$$\mathbf{A}_s^{\mathrm{T}} = \frac{1}{2ab}\begin{bmatrix}
-b(1-\eta) & -a(1-\xi) & w_1 \\
+b(1-\eta) & -a(1+\xi) & w_2 \\
+b(1+\eta) & +a(1+\xi) & w_3 \\
-b(1+\eta) & +a(1-\xi) & w_4 \\
\frac{ab}{2}(1-\eta) & 0 & \alpha_1 \\
\frac{ab}{2}(1-\eta) & 0 & \alpha_2 \\
\frac{ab}{2}(1+\eta) & 0 & \alpha_3 \\
\frac{ab}{2}(1+\eta) & 0 & \alpha_4 \\
0 & \frac{ab}{2}(1-\xi) & \beta_1 \\
0 & \frac{ab}{2}(1+\xi) & \beta_2 \\
0 & \frac{ab}{2}(1+\xi) & \beta_3 \\
0 & \frac{ab}{2}(1-\xi) & \beta_4
\end{bmatrix}. \tag{12.45}$$

The same interpolations are used for the virtual displacements. From the virtual work formula 12.32, the shear part of the stiffness matrix for the element is given by

$$\mathbf{k}_s = \int_A \mathbf{A}_s^{\mathrm{T}} \mathbf{C}_s \mathbf{A}_s \, \mathrm{d}x \, \mathrm{d}y$$

$$= \begin{matrix} \\ \mathbf{w} \\ \boldsymbol{\alpha} \\ \boldsymbol{\beta} \end{matrix} \begin{matrix} \begin{matrix} \mathbf{w} & \boldsymbol{\alpha} & \boldsymbol{\beta} \end{matrix} \\ \begin{bmatrix} \mathbf{k}_s^{11} & \mathbf{k}_s^{12} & \mathbf{k}_s^{13} \\ \mathbf{k}_s^{21} & \mathbf{k}_s^{22} & \mathbf{k}_s^{23} \\ \mathbf{k}_s^{31} & \mathbf{k}_s^{32} & \mathbf{k}_s^{33} \end{bmatrix} \end{matrix}, \tag{12.46}$$

where

$$\mathbf{k}_s^{11} = \frac{Ehk}{48(1+\nu)ab} \begin{bmatrix} 8a^2+8b^2 & 4a^2-8b^2 & -4a^2-4b^2 & -8a^2+4b^2 \\ 4a^2-8b^2 & 8a^2+8b^2 & -8a^2+4b^2 & -4a^2-4b^2 \\ -4a^2-4b^2 & -8a^2+4b^2 & 8a^2+8b^2 & 4a^2-8b^2 \\ -8a^2+4b^2 & -4a^2-4b^2 & 4a^2-8b^2 & 8a^2+8b^2 \end{bmatrix}, \tag{12.47}$$

$$\mathbf{k}_s^{21} = \frac{Ehkb}{48(1+\nu)} \begin{bmatrix} -4 & 4 & 2 & -2 \\ -4 & 4 & 2 & -2 \\ -2 & 2 & 4 & -4 \\ -2 & 2 & 4 & -4 \end{bmatrix} = (\mathbf{k}_s^{12})^{\mathrm{T}}, \tag{12.48}$$

$$\mathbf{k}_s^{21} = \frac{Ehkb}{48(1+\nu)} \begin{bmatrix} -4 & 4 & 2 & -2 \\ -4 & 4 & 2 & -2 \\ -2 & 2 & 4 & -4 \\ -2 & 2 & 4 & -4 \end{bmatrix} = \left(\mathbf{k}_s^{12}\right)^{\mathrm{T}}, \tag{12.49}$$

$$\mathbf{k}_s^{31} = \frac{Ehka}{48(1+\nu)} \begin{bmatrix} -4 & -2 & 2 & 4 \\ -2 & -4 & 4 & 2 \\ -2 & -4 & 4 & 2 \\ -4 & -2 & 2 & 4 \end{bmatrix} = (\mathbf{k}_s^{13})^{\mathrm{T}}, \tag{12.50}$$

$$\mathbf{k}_s^{22} = \frac{Ehkab}{48(1+\nu)} \begin{bmatrix} 2 & 2 & 1 & 1 \\ 2 & 2 & 1 & 1 \\ 1 & 1 & 2 & 2 \\ 1 & 1 & 2 & 2 \end{bmatrix}, \tag{12.51}$$

$$\mathbf{k}_s^{33} = \frac{Ehkab}{48(1+\nu)} \begin{bmatrix} 2 & 1 & 1 & 2 \\ 1 & 2 & 2 & 1 \\ 1 & 2 & 2 & 1 \\ 2 & 1 & 1 & 2 \end{bmatrix}, \tag{12.52}$$

$$\mathbf{k}_s^{32} = \mathbf{k}_s^{23} = \mathbf{0}. \tag{12.53}$$

For the analysis of shells, the plate bending stiffness must be combined with the membrane stiffness that can be derived from the plane stress analysis discussed in Chapter 5. We also need to introduce rotation about the z axis as an additional degree of freedom that alters the matrices.[4]

12.6 CLASSICAL PLATE THEORY

In the limiting case of very thin plates, the shears γ_{yz} and γ_{zx} are very small and can be neglected in the last two equations of 12.4. Thus, for theory of very thin plates,

$$\begin{aligned} \beta &= -\frac{\partial w}{\partial y}, \\ \alpha &= -\frac{\partial w}{\partial x}. \end{aligned} \tag{12.54}$$

Note that the shear strain is not zero, but it is small compared to the rotations. The formulas for γ_{xz} and γ_{yz} express the small difference of much larger quantities. There is then no constitutive relation for the shears, which become reactive quantities. This results in a reduction in the order of the governing differential equations. Combining 12.10, 12.14, and 12.54, one obtains the single differential equation

$$\nabla^4 w = \frac{p_z}{D}, \tag{12.55}$$

where

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}. \tag{12.56}$$

Since 12.55 is a fourth-order partial differential equation, the solution can satisfy only two boundary conditions on each edge. The appropriate conditions can be found from the virtual work formula. Using 12.54, from 12.25 the virtual work of the edge forces is

$$\int_C \left\{ \left(Q_x n_x + Q_y n_y \right) \bar{w} - (M_x n_x + M_{xy} n_y) \frac{\partial \bar{w}}{\partial x} - (M_{xy} n_x + M_y n_y) \frac{\partial \bar{w}}{\partial y} \right\} \mathrm{d}s. \quad (12.57)$$

On the side where x is constant, $n_y = 0$, $n_x = 1$, $\mathrm{d}s = \mathrm{d}y$, the virtual work of the edge forces is

$$\begin{aligned} &\int_{y_a}^{y_b} \left\{ Q_x \bar{w} - M_x \frac{\partial \bar{w}}{\partial x} - M_{xy} \frac{\partial \bar{w}}{\partial y} \right\} \mathrm{d}y \\ &= \int_{y_a}^{y_b} \left\{ Q_x \bar{w} - M_x \frac{\partial \bar{w}}{\partial x} - \frac{\partial}{\partial y} \left(M_{xy} \bar{w} \right) + \frac{\partial M_{xy}}{\partial y} \bar{w} \right\} \mathrm{d}y \qquad (12.58) \\ &= \int_{y_a}^{y_b} \left\{ \left(Q_x + \frac{\partial M_{xy}}{\partial y} \right) \bar{w} - M_x \frac{\partial \bar{w}}{\partial x} \right\} \mathrm{d}y - \left[M_{xy} \bar{w} \right]_{y_a}^{y_b}. \end{aligned}$$

Therefore, the appropriate boundary conditions are

$$\left.\begin{array}{l} w \text{ or } V_x \equiv Q_x + \dfrac{\partial Mxy}{\partial y} \\ \text{and} \\ \dfrac{\partial w}{\partial x} \text{ or } M_x \end{array}\right\} \text{given on the edge } x = \text{constant}, \quad (12.59)$$

and

$$\left.\begin{array}{l} w \text{ or } V_y \equiv Q_y + \dfrac{\partial Mxy}{\partial x} \\ \text{and} \\ \dfrac{\partial w}{\partial y} \text{ or } M_y \end{array}\right\} \text{given on the edge } y = \text{constant}. \quad (12.60)$$

This classical plate theory (CPT) does not allow separate boundary conditions on twisting moment and twisting rotation at a boundary. The combinations V_x and V_y are called the effective shear resultants. The last term in 12.58 is interpreted as a concentrated force of magnitude M_{xy} at the corner. A simply supported edge is modeled by zero displacement and zero bending moment. At the corner where two simply supported edges meet, there is a concentrated force $R = 2M_{xy}$ according to the CPT.

12.7 PLATE BENDING EXAMPLE

Only a few solutions of the system of Equations 12.22 and 12.23 are known. One important situation that can be analyzed is the case of a rectangular plate with two opposite edges simply supported with zero twisting restraint (simply supported type I). To construct a solution for this problem, it is convenient to introduce the following functions.[5]

$$\Phi = \frac{\partial \alpha}{\partial x} + \frac{\partial \beta}{\partial y},$$
$$\Psi = \frac{\partial \beta}{\partial x} - \frac{\partial \alpha}{\partial y}. \tag{12.61}$$

From 12.22 and 12.23,

$$\nabla^2 \Phi + \frac{p_z}{D} = 0 \tag{12.62}$$

and

$$\frac{\partial w}{\partial x} = -\alpha + \frac{D}{Ghk}\left(\frac{\partial \Phi}{\partial x} - \frac{1-\nu}{2}\frac{\partial \psi}{\partial y}\right),$$
$$\frac{\partial w}{\partial y} = -\beta + \frac{D}{Ghk}\left(\frac{\partial \Phi}{\partial y} + \frac{1-\nu}{2}\frac{\partial \psi}{\partial x}\right). \tag{12.63}$$

From which it follows that

$$\nabla^2 w = -\Phi - \frac{p_z}{Gkh}.$$
$$\Psi - \frac{h^2}{10}\nabla^2 \Psi = 0, \tag{12.64}$$

where we have used $k = 5/6$. Once Equations 12.62 and 12.64 are solved, the plate bending deformations and stress resultants can be calculated.

As an example, we consider a uniform load p on the rectangular plate with edge conditions as shown in Figure 12.3:

$$\left.\begin{aligned} w &= 0, \\ \beta &= 0, \\ M_x &= 0, \end{aligned}\right\} \text{on the edge } x = \text{constant}, \tag{12.65}$$

and

$$\left.\begin{aligned} w &= 0, \\ M_y &= 0, \\ M_{xy} &= 0, \end{aligned}\right\} \text{on the edge } y = \text{constant}, \tag{12.66}$$

corresponding to the two sets of conditions that may be called simply supported.

The uniform load of magnitude p can be represented by a Fourier sine series in the x direction:

$$p_z = \sum_n p_n \sin(\omega x), \quad p_n = \frac{4p}{n\pi}, \quad \omega = \frac{n\pi}{a}. \tag{12.67}$$

Then, a solution can be constructed by series expansions:

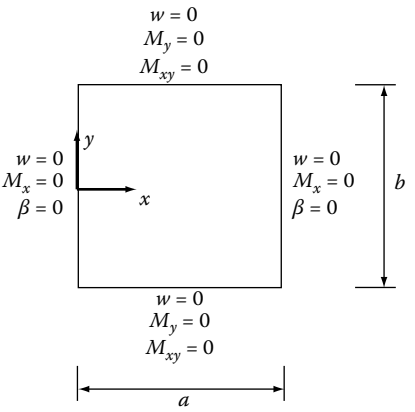


FIGURE 12.3 Plate with mixed boundary conditions.

$$w(x,y)=\sum_n W_n(y)\sin(\omega x),$$
$$\Phi(x,y)=\sum_n F_n(y)\sin(\omega x), \tag{12.68}$$
$$\Psi(x,y)=\sum_n G_n(y)\cos(\omega x).$$

Omitting the subscript n, we easily find a solution for each component of the load in terms of hyperbolic functions:

$$F=c_1\mathrm{Ch}(\omega y)+c_2\mathrm{Sh}(\omega y)+\frac{p_n}{D\alpha^2},$$
$$W=c_3\mathrm{Ch}(\omega y)+c_4\mathrm{Sh}(\omega y)-\frac{c_1}{2\omega}y\mathrm{Sh}(\omega y)-\frac{c_2}{2\omega}y\mathrm{Ch}(\omega y)+\frac{p_n}{D\omega^4}\left(1+\frac{\omega^2h^2}{5(1-\nu)}\right),$$
$$G=c_5\mathrm{Ch}(\gamma y)+c_6\mathrm{Sh}(\gamma y),\quad \gamma^2=\omega^2\left(1+\frac{10}{h^2\omega^2}\right). \tag{12.69}$$

Because of the symmetry about the x axis, we need to retain only the following terms:

$$F=c_1\mathrm{Ch}(\omega y)+\frac{p_n}{D\omega^2},$$
$$W=c_3\mathrm{Ch}(\omega y)-\frac{c_1}{2\omega}y\mathrm{Sh}(\omega y)+\frac{p_n}{D\omega^4}\left(1+\frac{\omega^2h^2}{5(1-\nu)}\right), \tag{12.70}$$
$$G=c_6\mathrm{Sh}(\gamma y).$$

The three constants are determined by the conditions on $y = 0$. The expressions are rather lengthy so I will not reproduce them here.

This problem was solved for $a = b$ by finite element method (FEM) (Section 15.26). We only need to use one-quarter of the plate with support conditions as shown in Figure 12.4. A uniform 20 × 20 mesh for the one-quarter was used. The calculated displacement at the center of the plate for $h/a = 0.1$, compared to the thin-plate approximation (CPT) and the exact solution of the shear theory using 400 terms in the series expansion, is shown in Table 12.1. The support reaction $q_x/(pa)$ on the edge $x = 0$ is shown in Figure 12.5. The support

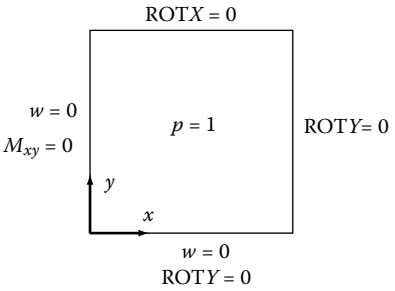


FIGURE 12.4 One-quarter of a plate with mixed boundary conditions.

TABLE 12.1
Center Displacement

	Exact	FEM	CPT
$\frac{wD}{pa^4}\times 10^6$	4456	4450	4060

reaction $q_y/(pa)$ on the edge $y = 0$, compared to the exact solution, is shown in Figure 12.6.

The finite element results are for the centroid of the element ($y/a = 0.0125$) and the exact solution is for $y = 0$. Note there is a large negative Q_y force near the corner. This is the accurate description of the reaction represented by the concentrated corner force in the CPT. In addition to the force Q_y, the support mechanism at this edge must provide a twisting moment $M_{xy}/(pa^2)$ as shown in Figure 12.7. The CPT

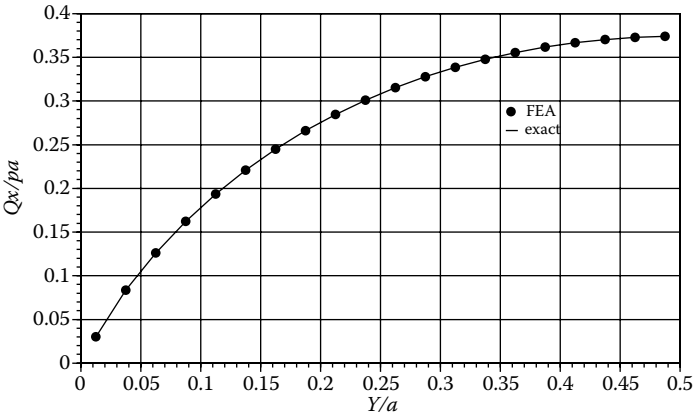


FIGURE 12.5 Shear resultant along edge with zero twisting moment.

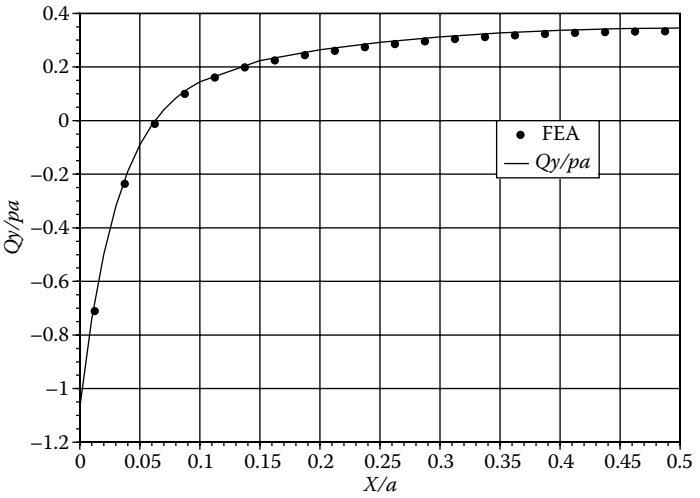


FIGURE 12.6 Shear resultant along edge with zero rotation.

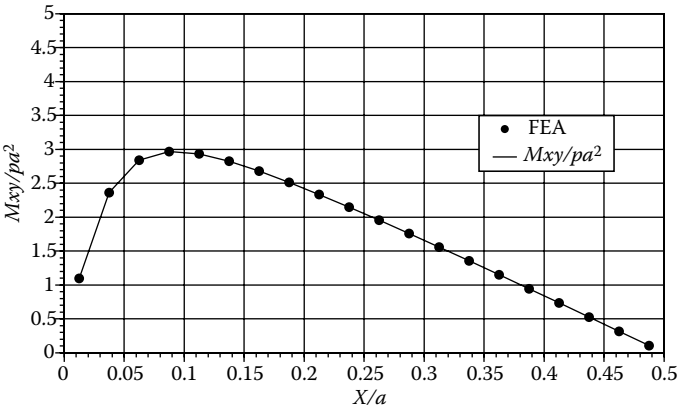


FIGURE 12.7 Twisting moment along edge with zero rotation.

resolves this combination of reactions into the effective shear reaction V_y and the corner reaction R.

Version 12 of ANSYS recommends the four-node element SHELL181 and the eight-node element SHELL281, which allow large deformations, plasticity, and viscoelasticity. Older versions of ANSYS offer several plate bending elements (43, 63, 93, 143). Table 12.2 shows the results using element SHELL181 for h/a =0.1 and a 20×20 mesh over one-quarter of the plate. Values of the stress resultants are at the element centroid. Maximum values are shown, except for the negative value of shear near the corner on the side with zero twisting rotation. SHELL181a uses the full integration option with incompatible modes. SHELL181b uses the reduced integration option.

TABLE 12.2
Comparison of Plate Elements

	SHELL181a at Centroid	SHELL181b at Centroid	EXACT at the Edge
$\frac{wD}{pa^4}\times 10^6$	4,451	4,450	4,456
$\frac{M_x}{pa^2}\times 10^5$	4,909	4,908	4,919
$\frac{M_y}{pa^2}\times 10^5$	4,971	4,970	
$\frac{M_{xy}}{pa^2}\times 10^5$	2,968	2,967	
$\frac{Q_x}{pa}\times 10^4$	3,740	3,740	
$\frac{Q_y}{pa}\times 10^4$	+3,331 −7,108	+3,331 −7,108	+3,455 −10,700

12.8 PROBLEMS

1. Derive the virtual work formula 12.25 for bending of plates. Note that

$$\int_A \frac{\partial f}{\partial x}\,\mathrm{d}a = \oint_c f n_x\,\mathrm{d}s, \quad \int_A \frac{\partial f}{\partial y}\,\mathrm{d}a = \oint_c f n_y\,\mathrm{d}s. \tag{12.71}$$

2. Use ANSYS, SHELL281, to determine the deflection at the center for a uniformly loaded square plate with clamped edges for $a = b = 20$, $h = 1$, $\nu = 0.3$, $D = 100$, so that $E = 10.92$. Use 1/4 of the plate and symmetry conditions. Submit element layout, maximum deflection, maximum and minimum values of moments and shears, and plot of QY for the clamped edge $y = 0$. See Section 15.27.
3. Solve the plate bending problem shown in Figure 12.8 using a 10 × 10 grid over one-quarter of the plate and symmetry conditions. See Section 15.26. Use $a = b = 10$, $h = 1$, $\nu = 0.3$, $D = 1$, so that $E = 10.92$. Submit:
   (a) Maximum UZ and the (x,y) location.
   (b) Maximum and minimum values of QY and (x,y) location.
   (c) Graph of QY along line $y = 0$.
4. Use ANSYS to analyze a cylindrical shell roof under gravity loading (Figure 12.9).

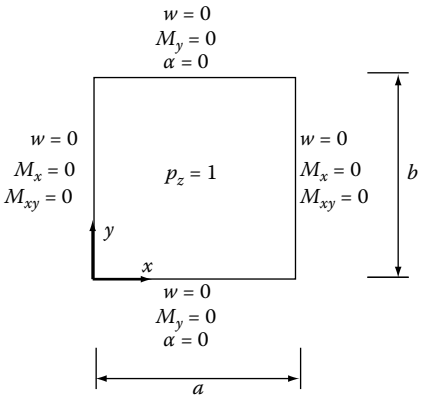


FIGURE 12.8 Plate boundary conditions.

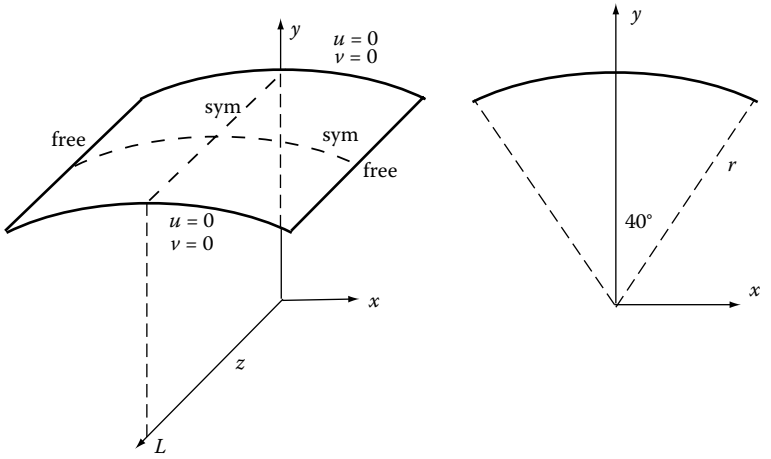


FIGURE 12.9 Shell roof.

The cylindrical shell is loaded by gravity in the y direction. The ends are supported by flexible membranes (walls) and the sides are free, $E = 3 \times 10^6$, $\nu = 0$, specific weight = 0.208 lb/in.^{3}, $r = 300$ in., thickness = 3 in., length = 600 in. Do a stress analysis of one-quarter of the shell using double symmetry. See Section 15.28.

REFERENCES

1. Bathe, K. J., MITC element, *Finite Element Procedures*, Prentice Hall, Englewood Cliffs, NJ, 1996, p. 425.
2. Hughes, T. J. R., T1 element, *The Finite Element Method*, Dover Pub., Mineola, NY, 2000, p. 342.

3. Zienkiewicz, O. C., and R. L. Taylor, *The Finite Element Method for Solid and Structural Mechanics*, Elsevier, New York, 2005, 6th ed., p. 397.
4. MacNeal, R. H., and R. L. Harder, A refined four-noded membrane element with rotational degrees of freedom, *Comput. Struct*., 28 (1), 75–84, 1988.
5. Marguerre, K., and H.-T. Woernle, *Elastic Plates*, Blaisdell Pub. Co., New York, 1969.

BIBLIOGRAPHY

Bathe, K. J., *Finite Element Procedures*, Prentice-Hall, Englewood Cliffs, NJ, 1996.
Cook, R. D., D. S. Malkus, M. E. Plesha, and R. J. Witt, *Concepts and Applications of Finite Element Analysis*, 4th ed., John Wiley & Sons, New York, 2002.

13 Large Deformations

13.1 THEORY OF LARGE DEFORMATIONS

Let us select one known configuration of the body as a reference configuration in which the body occupies a known region $\mathcal{R}$ of space.* This can be the configuration at any time. We will use a single rectangular Cartesian coordinate system. Let X_k denote the position of a particle in the reference configuration. In a deformed configuration $r(t)$ at time t, this particle occupies a position x_k. The motion relative to the fixed reference configuration is described by the mapping:

$$x_k = x_k(\mathbf{X},t). \tag{13.1}$$

The velocity of a particle is

$$v_k = \frac{\partial x_k}{\partial t}. \tag{13.2}$$

The acceleration of a particle is

$$a_k = \frac{\partial v_k}{\partial t}. \tag{13.3}$$

The tensor **G** with components

$$G_{km} = \frac{\partial x_k}{\partial X_m} \tag{13.4}$$

is called the deformation gradient.† The law of conservation of mass is

$$\frac{\rho_0}{\rho} = \left|G_{ij}\right| = J. \tag{13.5}$$

The balance of momentum applied to an element of the deformed body gives‡

* For a complete treatment of the general theory, see Dill: Chapter 2. Refer to Preface.

† We use here **G** for gradient in place of the more common **F**, which is reserved for force.

‡ Throughout this section, the summation convention applies to repeated indices and their range is 1, 2, 3.

$$\frac{\partial T_{km}}{\partial x_k} + \rho b_m = \rho a_m, \quad T_{km} = T_{mk}, \tag{13.6}$$

where T_{ij} are the components of the true (Cauchy) stress tensor. This formulation is not convenient for finite element calculations since the independent variables are the unknown coordinates x_k of a particle in the unknown deformed position. A change of variables leads to the more convenient form of balance of momentum:

$$\frac{\partial P_{mk}}{\partial X_k} + \rho_0 b_m = \rho_0 a_m, \tag{13.7}$$

where

$$P_{ji} = \frac{\rho_0}{\rho} G_{ik}^{-1} T_{kj}, \quad T_{rm} = \frac{\rho}{\rho_0} G_{ri} P_{mi}. \tag{13.8}$$

The new tensor **P** is usually called the first Piola–Kirchhoff tensor.* Note that the derivatives are now with respect to the coordinates of position in the reference configuration, which are known quantities.

13.1.1 Virtual Work

If we multiply 13.7 by arbitrary functions $\phi_m(\mathbf{X})$, called virtual displacements, and integrate over the reference configuration, we have†

$$\int_{\mathcal{R}} \left(\frac{\partial P_{mk}}{\partial X_k} + \rho_0 b_m - \rho_o a_m \right) \phi_m \, \mathrm{d}V = 0. \tag{13.9}$$

Integration by parts yields

$$\int_{\mathcal{R}} P_{mk} \frac{\partial \phi_m}{\partial X_k} \mathrm{d}V = \int_{\partial \mathcal{R}} p_m \phi_m \, \mathrm{d}A + \int_{\mathcal{R}} (\rho_0 b_m - \rho_0 a_m) \phi_m \, \mathrm{d}V, \tag{13.10}$$

where the vector

$$p_m = P_{mk} N_k \tag{13.11}$$

is the surface traction per unit area of the reference configuration and **N** is the unit normal to the reference surface. This gives a physical interpretation for the Piola ten-

* **P** is the transpose of the Piola tensor in my book on continuum mechanics.

† The more usual notation is $\phi_m = \delta x_m$.

sor $\mathbf{P}$: P_{ij} is the jth component of the traction per unit undeformed area acting on the deformed surface that was the ith coordinate surface in the undeformed body.

Equation 13.10 is the virtual work formula for large deformations. The balance of momentum is satisfied if and only if this relation holds for all smooth functions $\phi_m(\mathbf{X})$. In order to complete the theory, we need the specific constitutive relation for the material. We will consider here only elastic materials at uniform temperature.

13.1.2 Elastic Materials

An elastic material is one for which the stress in the deformed configuration depends only on the local deformation in that configuration and not on the history of deformation. Let the reference configuration be unstressed. The relation between stress and deformation must be such that the body remains stress-free in a rigid body motion. The general representation of such a relation is

$$T_{ij} = \frac{\rho}{\rho_0} G_{ik} f(\mathbf{C})_{km} G_{jm}, \tag{13.12}$$

where

$$C_{ij} = G_{ki} G_{kj} \tag{13.13}$$

is the deformation tensor. Let us introduce the new tensor

$$S_{ij} = P_{mi} G^{-1}_{jm} = J G^{-1}_{ik} T_{km} G^{-1}_{jm}, \quad P_{ki} = G_{kj} S_{ij}, \tag{13.14}$$

which I will call the Kirchhoff tensor.* The constitutive relation then has a simple form

$$S_{ij} = f(\mathbf{C})_{ij}. \tag{13.15}$$

Each function f defines a particular elastic material. We can show from thermodynamic considerations that f is the derivative of the strain energy with respect to the strain tensor $\mathbf{E}$:

$$2E_{ij} = C_{ij} - \delta_{ij}, \tag{13.16}$$

and†

$$S_{ij} = \frac{\partial W(\mathbf{C})}{\partial E_{ij}} = 2\frac{\partial W(\mathbf{C})}{\partial \mathrm{C}_{ij}}. \tag{13.17}$$

* $\mathbf{S}$ is what is often called the second Piola–Kirchhoff tensor; see Dill, p. 22. Refer to Preface.

† Since $E_{ij} = E_{ji}$, the variables are not independent and care must be taken in evaluating the partial derivatives; see Dill, p. 321. Refer to Preface.

In tensor notation,

$$\mathbf{S} = \frac{\partial W}{\partial \mathbf{E}} = 2\frac{\partial W}{\partial \mathbf{C}}. \tag{13.18}$$

Since **C** depends on the reference configuration, the form of the constitutive relation also depends on the choice of reference configuration.

If the material is isotropic, the strain energy depends only on the principal invariants of **C**:

$$I_1 = \text{tr}\,\mathbf{C},\ 2I_2 = I_1^2 - \text{tr}\mathbf{C}^2,\ I_3 = \det\mathbf{C} = J^2. \tag{13.19}$$

Their derivatives are

$$\frac{\partial I_1}{\partial \mathbf{C}} = \mathbf{1},\ \frac{\partial I_2}{\partial \mathbf{C}} = I_1\mathbf{1} - \mathbf{C},\ \frac{\partial I_3}{\partial \mathbf{C}} = I_3\mathbf{C}^{-1}. \tag{13.20}$$

For an initial state of zero strain, $\mathbf{C} = 1$ and therefore $I_1 = 3$, $I_2 = 3$, and $I_3 = 1$. Let $J_1 = I_1 - 3$, $J_2 = I_2 - 3$, and $J_3 = I_3 - 3$. Then, equivalently, $W = W(J_1,J_2,J_3)$ and we can represent any material by a power series:

$$W = \sum_{n,m,p=0}^{\infty} c_{nmp} J_1^n J_2^m J_3^p. \tag{13.21}$$

Since there is a one-to-one correspondence between the principal invariants and the principal stretches $\alpha_i = c_i^2$, we can also regard the free energy as a function of the principal stretches for an isotropic material.

$$W = W(\alpha_1, \alpha_2, \alpha_3), \tag{13.22}$$

where

$$W(\alpha_1, \alpha_2, \alpha_3)W = W(\alpha_2, \alpha_3, \alpha_1)W = W(\alpha_3, \alpha_1, \alpha_2). \tag{13.23}$$

A general formula is alternatively obtained by a power series in the principal stretches.[1]

From 13.18, the principal values of **S** are given by

$$S_i = 2\frac{\partial W}{\partial c_i} = \alpha_i^{-1}\frac{\partial W}{\partial \alpha_i} \quad \text{(no sum)}. \tag{13.24}$$

Since the principal axes of **C** and **S** coincide for an isotropic material, this relation completely defines the constitutive relation. The principal stresses are

$$T_i = J^{-1}\alpha_i \frac{\partial W}{\partial \alpha_i}, \quad \text{(no sum)}. \tag{13.25}$$

Since the sequence of partial derivatives can be exchanged, allowable expressions for the stress tensor are restricted by the relation

$$\frac{\partial}{\partial \alpha_j}\left(\frac{JT_i}{\alpha_i}\right) = \frac{\partial}{\partial \alpha_i}\left(\frac{JT_j}{\alpha_j}\right) \quad \text{(no sum)}. \tag{13.26}$$

To decide whether a specific model accurately characterizes a particular real material, one must compare the predictions of the model with results of some laboratory tests on the material. The most common test for solids is the tensile test.

Let us consider a tensile test of a square rod. The x_1 axis is in the direction of loading. If l is the deformed length and l_0 is the original length, the longitudinal *stretch* is $\alpha_1 = l/l_0$ and the *extension* is $\delta = \alpha_1 - 1 = (l - l_0)/l_0$. The extension is also called the "normal strain" or "engineering strain." If a is the deformed width and a_0 is the original width, the transverse stretch is $\alpha_2 = \alpha_3 = a/a_0$. The area of the original cross section is $A_0 = a_0^2$. The area of the cross section of the deformed rod is $A = a^2$. The ratio of the areas is therefore $A/A_0 = \alpha_2\alpha_3$.

The deformation is described by

$$x_1 = \alpha_1 X_1, \; x_2 = \alpha_2 X_2, \; x_3 = \alpha_3 X_3. \tag{13.27}$$

The nonzero components of the deformation gradient are

$$G_{11} = \alpha_1, \, G_{22} = \alpha_2, \, G_{33} = \alpha_3. \tag{13.28}$$

The nonzero components of the deformation tensor are

$$C_{11} = \alpha_1^2, \quad C_{22} = \alpha_2^2, \quad C_{33} = \alpha_3^2. \tag{13.29}$$

The invariants of $\mathbf{C}$ are

$$\begin{aligned} I_1 &= \alpha_1^2 + \alpha_2^2 + \alpha_3^2, \\ I_2 &= \alpha_1^2\alpha_2^2 + \alpha_2^2\alpha_3^2 + \alpha_3^2\alpha_1^2, \\ I_3 &= \alpha_1^2\alpha_2^2\alpha_3^2. \end{aligned} \tag{13.30}$$

If F is the resultant force on the cross section, we have

$$P_{11} = \frac{F}{A_o}, \quad T_{11} = \frac{P_{11}}{\alpha_2\alpha_3}, \quad S_{11} = \frac{P_{11}}{\alpha_1}. \tag{13.31}$$

P_{11} is called the "nominal stress" or "engineering stress." T_{11} is the true stress.

Experimental measurements provide P_{11}, α_1, α_2, and α_3 as a function of the extension δ_1. For an isotropic material, the constitutive relations 13.15 have the form

$$\begin{aligned} S_{11} &= f\left(\alpha_1,\alpha_2,\alpha_3\right), \\ S_{22} &= f\left(\alpha_2,\alpha_3,\alpha_1\right), \\ S_{33} &= f\left(\alpha_3,\alpha_1,\alpha_2\right). \end{aligned} \tag{13.32}$$

For the tensile test of an isotropic material, $\alpha_2 = \alpha_3$ and $S_{22} = S_{33} = 0$. For measured values at times t_n, $n = 1$ to N, we have

$$\begin{aligned} f\left(\alpha_1(t_n),\alpha_2(t_n),\alpha_2(t_n)\right) &= S_{11}(t_n), \\ f\left(\alpha_2(t_n),\alpha_2(t_n),\alpha_1(t_n)\right) &= 0. \end{aligned} \tag{13.33}$$

The adopted constitutive relation must contain sufficient undetermined parameters so that relation 13.32 can fit the experimental data with sufficient accuracy. We will review some models below.

To separate the distortional strains from the volumetric strains, we will introduce as a measure of deformation[2]

$$\bar{\mathbf{C}} = J^{-1/3}\mathbf{C}, \tag{13.34}$$

so that

$$\left|\bar{\mathbf{C}}\right| = 1. \tag{13.35}$$

The corresponding principal stretches are

$$\bar{\alpha}_i = J^{-1/3}\alpha_i. \tag{13.36}$$

Note that

$$\bar{\alpha}_1\bar{\alpha}_2\bar{\alpha}_3 = 1. \tag{13.37}$$

The principal invariants of $\mathbf{C}$ and $\bar{\mathbf{C}}$ are related by

$$\begin{aligned} \bar{I}_1 &= I_1 I_3^{-1/3}, \\ \bar{I}_2 &= I_2 I_3^{-2/3}, \\ J &= I_3^{1/2}. \end{aligned} \tag{13.38}$$

We can regard the strain energy as a function of $(\bar{I}_1,\bar{I}_2,J)$ or of $(\bar{\alpha}_1,\bar{\alpha}_2,\bar{\alpha}_3,J)$.

For a purely volumetric deformation $\alpha_1 = \alpha_2 = \alpha_3 \equiv \alpha$, $J = \alpha^3$, so that $\bar{I}_1 = \bar{I}_2 = 3$. This suggests the use of the invariants

$$\bar{J}_1 = \bar{I}_1 - 3, \quad \bar{J}_2 = \bar{I}_2 - 3, \quad \bar{J}_3 = J - 1, \tag{13.39}$$

and expressing

$$W = W(\bar{J}_1, \bar{J}_2, \bar{J}_3). \tag{13.40}$$

Examples are given below.

Any relation that applies to large deformations must also apply to small deformation gradients. From 13.4, we find*

$$\mathbf{G} = \mathbf{1} + \mathbf{H} \tag{13.41}$$

where $\mathbf{H}$ is the displacement gradient. For small deformations

$$\|\mathbf{H}\| = O(\varepsilon) \tag{13.42}$$

where $\varepsilon << 1$. The small strain tensor is defined by

$$\boldsymbol{\varepsilon} = \frac{1}{2}\left(\mathbf{H} + \mathbf{H}^{\mathrm{T}}\right). \tag{13.43}$$

Up to first order,

$$\begin{aligned} &\mathbf{C} \doteq \mathbf{1} + 2\boldsymbol{\varepsilon}, \\ &\mathbf{E} = \frac{1}{2}(\mathbf{C} - \mathbf{1}) \doteq \boldsymbol{\varepsilon}, \\ &\mathbf{P} \doteq \mathbf{S} \doteq \mathbf{T}. \end{aligned} \tag{13.44}$$

For the tensile test, $\alpha_{\mathrm{i}} = 1 + \varepsilon_{\mathrm{i}}$, where $\varepsilon_i << 1$, $\alpha_i^2 \doteq 1 + 2\varepsilon_i$, etc. The invariants of $\mathbf{C}$ are approximately

$$I_1 \doteq 3 + 2\mathrm{tr}\,\varepsilon, \quad I_2 \doteq 3 + 4\mathrm{tr}\,\boldsymbol{\varepsilon}, \quad I_3 \doteq 1 + 2\mathrm{tr}\,\boldsymbol{\varepsilon}. \tag{13.45}$$

And $\mathrm{tr}\,\boldsymbol{\varepsilon} = \varepsilon_1 + \varepsilon_2 + \varepsilon_3$ is the volumetric strain.

13.1.3 Mooney–Rivlin Model of an Incompressible Material

In this case, the material is modeled as *incompressible*, that is, $\rho = \rho_0$ and the mean stress is determined by that constraint, rather than a constitutive relation. The Mooney–Rivlin relation is most compactly stated directly in terms of the true stress tensor. With respect to a stress-free reference configuration,

* We use here the standard matrix notation: for example, $\mathbf{G} = [G_{km}]$, where k is the row index and m is the column index.

$$T_{ij} = -p\delta_{ij} + 2c_1 B_{ij} - 2c_2 B_{ij}^{-1}, \tag{13.46}$$

where

$$B_{ij} = G_{ik}G_{jk}, \tag{13.47}$$

and p is an undetermined parameter that has to be found by solving the boundary value problem. The material is isotropic. For the tensile test,

$$B_{11} = \alpha_1^2, \quad B_{22} = \alpha_2^2, \quad B_{33} = \alpha_3^2, \tag{13.48}$$

and the other components are zero. Therefore,

$$\begin{aligned} T_{11} &= -p + 2c_1\alpha_1^2 - 2c_2\alpha_1^{-2}, \\ T_{22} &= -p + 2c_1\alpha_2^2 - 2c_2\alpha_2^{-2}, \\ T_{33} &= -p + 2c_1\alpha_3^2 - 2c_2\alpha_3^{-2}. \end{aligned} \tag{13.49}$$

The condition $T_{22} = T_{33} = 0$ shows that $\alpha_2 = \alpha_3$ and determines p as a function of the stretches. The condition of incompressibility is $\alpha_1\alpha_2\alpha_3 = 1$. Therefore,

$$\alpha_2 = \alpha_3 = \alpha_1^{-1/2} \tag{13.50}$$

and the axial stress is

$$T_{11} = 2c_1\left(\alpha_1^2 - \alpha_1^{-1}\right) - 2c_2\left(\alpha_1^{-2} - \alpha_1\right). \tag{13.51}$$

The nominal (Piola) stress, is

$$P_{11} = \alpha_2\alpha_3 T_{11} = \alpha_1^{-1}T_{11}. \tag{13.52}$$

Plots of load (nominal stress) versus extension and (true) stress versus extension are shown in Figures 13.1 and 13.2 for $c_1 = c_2$. In tension, the stress is nearly proportional to the extension, but the proportionality does not hold for large compressions.

13.1.4 Generalized Mooney–Rivlin Model

The Mooney–Rivlin model[3,4] can be extended to allow for compressibility of the material by using the variables in 13.39 and the formula for the strain energy 13.40 in the form

$$W = a\bar{J}_1 + b\bar{J}_2 + \frac{\kappa}{2}\bar{J}_3^2. \tag{13.53}$$

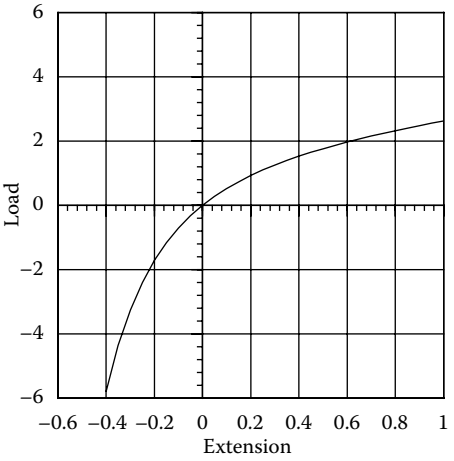


FIGURE 13.1 Load extension for tensile test of a Mooney–Rivlin material.

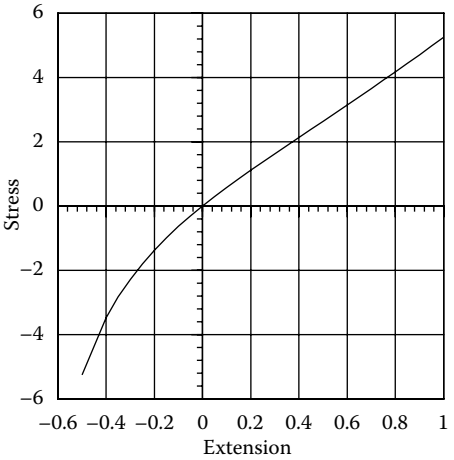


FIGURE 13.2 Stress extension for tensile test of a Mooney–Rivlin material.

From 13.17,

$$\mathbf{S} = a\mathbf{A}_1 + b\mathbf{A}_2 + \kappa(J-1)J\mathbf{C}^{-1} \tag{13.54}$$

where

$$\begin{aligned}\mathbf{A}_1 &= 2\frac{\partial \bar{I}_1}{\partial \mathbf{C}} = 2J^{-2/3}\left(\mathbf{1} - \frac{1}{3}\bar{I}_1\bar{\mathbf{C}}^{-1}\right),\\ \mathbf{A}_2 &= 2\frac{\partial \bar{I}_2}{\partial \mathbf{C}} = 2J^{-2/3}\left(\bar{I}_1\mathbf{1} - \bar{\mathbf{C}} - \frac{2}{3}\bar{I}_2\bar{\mathbf{C}}^{-1}\right).\end{aligned} \tag{13.55}$$

The stress tensor is given by

$$J\mathbf{T} = 2a\bar{\mathbf{B}} - 2b\bar{\mathbf{B}}^{-1} + \left(\kappa(J-1)J - \frac{1}{3}2a\bar{I}_1 + \frac{1}{3}2b\bar{I}_2\right)\mathbf{1}, \tag{13.56}$$

where

$$\bar{\mathbf{B}} = J^{-2/3}\mathbf{B}. \tag{13.57}$$

If the material is incompressible, $J = 1$ and the stress–strain relation reduces to the incompressible form 13.46.

The principal stresses are

$$JT_k = 2a_1\left(\bar{\alpha}_k^2 - \frac{1}{3}\bar{I}_1\right) - 2a_2\left(\bar{\alpha}_k^{-2} - \frac{1}{3}\bar{I}_2\right) + \kappa(J-1). \tag{13.58}$$

The volume change is modeled by the simple relation

$$p \equiv \frac{1}{3}\operatorname{tr}\mathbf{T} = \kappa(J-1). \tag{13.59}$$

For the tensile test, loading along the x_1 axis, $\bar{\alpha}_2 = \bar{\alpha}_3$ and condition 13.37 gives $\bar{\alpha}_2 = \bar{\alpha}_1^{-1/2}$. From 13.58,

$$JT_1 = 2a\left(\bar{\alpha}_1^2 - \bar{\alpha}_1^{-1}\right) + 2b\left(\bar{\alpha}_1 - \bar{\alpha}_1^{-2}\right). \tag{13.60}$$

Equations 13.59 and 13.60 determine α_1 and α_2 for a given T_1. For $a = b = 25$ and $\kappa = 500$, the solution is given in Table 13.1.

Figure 13.3 shows the stress–strain curve for the nominal stress, P_1/a, versus the extension $\alpha_1 - 1$.

TABLE 13.1
Mooney–Rivlin Model

T_1	α_1	α_2	P_1	J	$\bar{\alpha}_1$
10	1.0364	0.9855	9.713	1.00667	1.03411
20	1.0744	0.9711	18.862	1.01333	1.06973
30	1.1141	0.9568	27.465	1.02000	1.10680
40	1.1554	0.9427	35.545	1.02667	1.14527
50	1.1980	0.9287	43.126	1.03333	1.18502

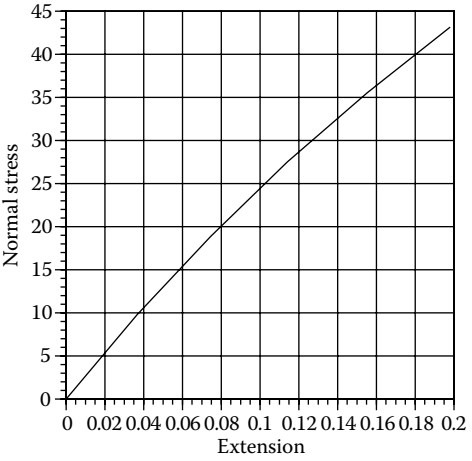


FIGURE 13.3 Tensile test of Mooney–Rivlin compressible material.

For small deformations, one finds

$$\boldsymbol{\tau} = 2(2a+2b)\boldsymbol{\varepsilon} + \left(\kappa - \frac{1}{3}(2a+2b)\right)(\mathrm{tr}\boldsymbol{\varepsilon})\mathbf{1}. \tag{13.61}$$

Therefore,

$$2(a+b) = \mu, \text{ and } \kappa = \lambda + \frac{2}{3}\mu. \tag{13.62}$$

That is, κ is the bulk modulus and μ is the shear modulus as measured for small deformations. We can write

$$a = \frac{1}{4}\left(\mu + c\right), \quad b = \frac{1}{4}\left(\mu - c\right). \tag{13.63}$$

There is then just one constant c to model the nonlinear behavior. We can fit only one data point on a measured nonlinear stress–strain curve. It is also possible to make an approximate fit to multiple measured points (see Section 13.2).

13.1.5 Polynomial Formula

The general representation 13.40 can be expanded in a power series. In doing so, we attempt to separate the volumetric and distortional strains by assuming a separation of the variable $\overline{J}_3$ from $(\overline{J}_1, \overline{J}_2)$:

$$W = \sum_{n=m=0}^{N} c_{nm} \overline{J}_1^{\,n} \overline{J}_2^{\,m} + \sum_{n=1}^{M} \frac{1}{2}\kappa_n \overline{J}_3^{\,2n}. \tag{13.64}$$

From 13.18,

$$\begin{aligned}\mathbf{S} &= \sum_{n=1}^{N} c_{n0} n \bar{J}_1^{n-1} \mathbf{A}_1 + \sum_{n=1}^{N} c_{0n} n \bar{J}_2^{n-1} \mathbf{A}_2 \\ &+ \sum_{n=1}^{N}\sum_{m=1}^{N} c_{nm} \left(n \bar{J}_1^{n-1} \bar{J}_2^{m} \mathbf{A}_1 + m \bar{J}_1^{n} \bar{J}_2^{m-1} \mathbf{A}_2 \right) \\ &+ \sum_{n=1}^{M} n \kappa_n \bar{J}_3^{2n-1} J \mathbf{C}^{-1},\end{aligned} \tag{13.65}$$

where $\mathbf{A}_1$ and $\mathbf{A}_2$ are defined by 13.55. For $N = 3$ and $M = 1$,

$$\begin{aligned}\mathbf{S} &= \left(c_{10} + c_{11}\bar{J}_2 + 2c_{20}\bar{J}_1 + 3c_{30}\bar{J}_1^2 + 2c_{21}\bar{J}_1\bar{J}_2 + c_{12}\bar{J}_2^2 \right) \mathbf{A}_1 \\ &+ \left(c_{01} + c_{11}\bar{J}_1 + 2c_{02}\bar{J}_2 + 3c_{03}\bar{J}_2^2 + 2c_{12}\bar{J}_1\bar{J}_2 + c_{21}\bar{J}_1^2 \right) \mathbf{A}_2 \\ &+ \kappa \left(J - 1 \right) J \mathbf{C}^{-1}.\end{aligned} \tag{13.66}$$

From 13.14, the true stress is

$$\begin{aligned}J\mathbf{T} &= \sum_{n=1}^{N} c_{n0} n \bar{J}_1^{n-1} \bar{\mathbf{A}}_1 + \sum_{n=1}^{N} c_{0n} n \bar{J}_2^{n-1} \bar{\mathbf{A}}_2 \\ &+ \sum_{n=1}^{N}\sum_{m=1}^{N} c_{nm} \left(n \bar{J}_1^{n-1} \bar{J}_2^{m} \bar{\mathbf{A}}_1 + m \bar{J}_1^{n} \bar{J}_2^{m-1} \bar{\mathbf{A}}_2 \right) \\ &+ \sum_{n=1}^{M} n \kappa_n \bar{J}_3^{2n-1} J \mathbf{1},\end{aligned} \tag{13.67}$$

where

$$\begin{aligned}\bar{\mathbf{A}}_1 &= 2\bar{\mathbf{B}} - \frac{2}{3}\mathbf{1}, \quad \bar{\mathbf{B}} = J^{-2/3}\mathbf{B}, \\ \bar{\mathbf{A}}_2 &= -2\bar{\mathbf{B}}^{-1} + \frac{2}{3}\bar{I}_2\mathbf{1}.\end{aligned} \tag{13.68}$$

For a purely volumetric strain: $\alpha_1 = \alpha_2 = \alpha_3 = \alpha$, $J = \alpha^3$, $\bar{J}_1 = \bar{J}_2 = 0$. In this case, from 13.67,

$$\frac{1}{3}\operatorname{tr}\mathbf{T} = \sum_{n=1}^{M} n \kappa_n (J-1)^{2n-1}. \tag{13.69}$$

The principal stress and principal stretches are related by

$$\hat{T}_1 - \hat{T}_2 = \sum_{n=1}^{N} 2c_{n0} n \bar{J}_1^{n-1} \left(\bar{\alpha}_1^2 - \bar{\alpha}_2^2 \right) + \sum_{n=1}^{N} 2c_{0n} n \bar{J}_2^{n-1} \left(\bar{\alpha}_2^{-2} - \bar{\alpha}_1^{-2} \right) + \sum_{n=1}^{N} \sum_{m=1}^{N} 2c_{nm} \left(n \bar{J}_1^{n-1} \bar{J}_2^{m} \left(\bar{\alpha}_1^2 - \bar{\alpha}_1^2 \right) + m \bar{J}_1^{n} \bar{J}_2^{m-1} \left(\bar{\alpha}_2^{-2} - \bar{\alpha}_1^{-2} \right) \right), \tag{13.70}$$

$$\hat{T}_2 - \hat{T}_3 = \sum_{n=1}^{N} 2c_{n0} n \bar{J}_1^{n-1} \left(\bar{\alpha}_2^2 - \bar{\alpha}_3^2 \right) + \sum_{n=1}^{N} 2c_{0n} n \bar{J}_2^{n-1} \left(\bar{\alpha}_3^{-2} - \bar{\alpha}_2^{-2} \right) + \sum_{n=1}^{N} \sum_{m=1}^{N} 2c_{nm} \left(n \bar{J}_1^{n-1} \bar{J}_2^{m} \left(\bar{\alpha}_2^2 - \bar{\alpha}_3^2 \right) + m \bar{J}_1^{n} \bar{J}_2^{m-1} \left(\bar{\alpha}_3^{-2} - \bar{\alpha}_2^{-2} \right) \right), \tag{13.71}$$

$$\bar{\alpha}_1 \bar{\alpha}_2 \bar{\alpha}_3 = 1. \tag{13.72}$$

where $\hat{\mathbf{T}} = J\mathbf{T}$ is the weighted, or Kirchhoff, stress measure.

In terms of the constants c_{nm}, the case $N = 1$ is known as the two-element model, $N = 2$ as the five-element model, and $N = 3$ as the nine-element model. The two-element model with one volume term is identical to the compressible Mooney–Rivlin model (13.53). For the two-element model,

$$\frac{1}{3} \operatorname{tr} \mathbf{T} = \kappa (J - 1) \tag{13.73}$$

for volumetric strain, and

$$\hat{T}_1 - \hat{T}_2 = 2c_{10} \left(\bar{\alpha}_1^2 - \bar{\alpha}_2^2 \right) + 2c_{01} \left(\bar{\alpha}_2^{-2} - \bar{\alpha}_1^{-2} \right), \tag{13.74}$$

$$\hat{T}_2 - \hat{T}_3 = 2c_{10} \left(\bar{\alpha}_2^2 - \bar{\alpha}_3^2 \right) + 2c_{01} \left(\bar{\alpha}_3^{-2} - \bar{\alpha}_2^{-2} \right), \tag{13.75}$$

for distortion.

13.1.6 Ogden's Function

Using the reduced principal stretches 13.36, a possible polynomial expansion is Ogden's potential:

$$W = \sum_{i=1}^{N} \frac{\mu_i}{a_i} \left(\bar{\alpha}_1^{a_i} + \bar{\alpha}_2^{a_i} + \bar{\alpha}_3^{a_i} - 3 \right) + \sum_{k=1}^{N} \frac{\kappa_k}{2} \left(J - 1 \right)^{2k} \tag{13.76}$$

where μ_i, a_i, and κ_k are the experimentally determined material constants. Applying this relation to the case of small deformations, we find that the shear modulus $\mu = \sum_{i=1}^{N} a_i \mu_i$ and κ_1 is the bulk modulus. The case $a_1 = 2$ and $a_2 = -2$ leads back to the Mooney–Rivlin model. Since $\bar{\alpha}_1 \bar{\alpha}_2 \bar{\alpha}_3 = 1$,

$$\bar{I}_2 = \bar{\alpha}_1 \bar{\alpha}_2 + \bar{\alpha}_2 \bar{\alpha}_3 + \bar{\alpha}_3 \bar{\alpha}_1 = \bar{\alpha}_1^{-2} + \bar{\alpha}_2^{-2} + \bar{\alpha}_3^{-2}. \tag{13.77}$$

Using only the first two terms of the series and one volume term, 13.25 gives

$$T_k = \mu_1 J^{-5/3} \left(\alpha_k^2 - \frac{1}{3} I_1 \right) - \mu_2 J^{-1/3} \left(\alpha_k^{-2} - \frac{1}{3} I_2 I_3^{-1} \right) + \kappa_1 (J - 1). \tag{13.78}$$

The small deformation shear modulus $\mu = \mu_1 + \mu_2$ and κ_1 is the bulk modulus.

13.1.7 Blatz–Ko Model

The following constitutive relation for isothermal deformations of an isotropic elastic material was originally devised from experiments on polyurethane foam.[5,6] The formula for the strain energy is

$$W = \frac{\mu}{2} \left(\frac{I_2}{I_3} + 2\sqrt{I_3} - 5 \right). \tag{13.79}$$

Using 13.20, the stress–strain relations is found to be

$$\mathbf{S} = \mu \left(\frac{I_1}{I_3} \mathbf{1} - \frac{1}{I_3} \mathbf{C} + \left(\sqrt{I_3} - \frac{I_2}{I_3} \right) \mathbf{C}^{-1} \right). \tag{13.80}$$

For small deformations, using 13.45 and $\mathbf{S} \doteq \boldsymbol{\tau}$, the stress–strain relation is

$$\boldsymbol{\tau} = 2\mu\varepsilon + \mu(\text{tr } \boldsymbol{\varepsilon})\mathbf{1}. \tag{13.81}$$

We see that μ is the shear modulus and the Poisson ratio is $\nu = 1/4$, that is, $\lambda = \mu$.

To illustrate the behavior of this material, let us consider the tensile test. Using 13.29,

$$S_i = \mu I_3^{-1} \left(I_1 - \alpha_i^2 - I_2 \alpha_i^{-2} + I_3^{3/2} \alpha_i^{-2} \right). \tag{13.82}$$

For a tensile test along the x axis, $S_2 = S_3 = 0$, and we find that

$$\alpha_2 = \alpha_3, \alpha_2^4 = \alpha_1^{-1}. \tag{13.83}$$

It then follows that

$$S_1 = \mu\left(\alpha_1^{-3/2} - \alpha_1^{-4}\right). \tag{13.84}$$

The true stress is

$$T_{11} = \mu\left(1 - \alpha_1^{-5/2}\right). \tag{13.85}$$

The nominal stress (Piola tensor) is

$$P_{11} = \mu\left(\alpha_1^{-1/2} - \alpha_1^{-3}\right). \tag{13.86}$$

Some calculated values are shown in Table 13.2.

The results are plotted in Figure 13.4, where G denotes the shear modulus μ. The material is not stable for large stretch: $dS_{11}/d\alpha_1 = 0$ at $\alpha_1 = 1.48$. The model cannot

TABLE 13.2
Blatz–Ko Tensile Test

$\alpha_1 - 1$	P_{11}/μ
0	0
0.1	0.20215
0.2	0.33417
0.3	0.42189

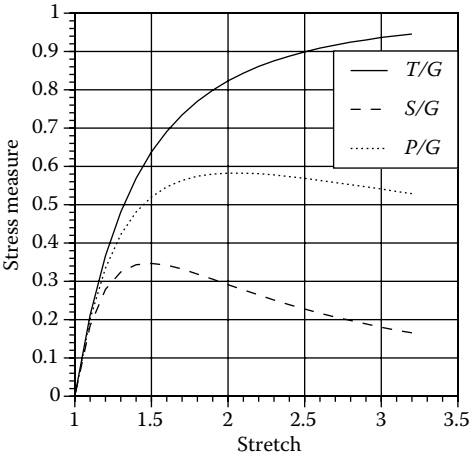


FIGURE 13.4 Stress vs. stretch for tensile test of Blatz–Ko model.

apply to a real material for greater stretches, and one can expect computational difficulties if the Piola (nominal) stress approaches one-half of the shear modulus, or any stretch ratio approaches 1.5.

13.1.8 Logarithmic Strain Measure

The logarithmic strain measure is defined as

$$\mathbf{e} = \ln(\mathbf{V}) = \sum_{i=1}^{3} \ln(\alpha_i)\mathbf{n}_i\mathbf{n}_i \tag{13.87}$$

where $(\alpha_i, \mathbf{n}_i)$ are the eigenvalues and eigenvectors of the left stretch tensor $\mathbf{V} = \mathbf{G}\mathbf{G}^{\mathrm{T}}$. The constitutive relation

$$\mathbf{T} = 2\mu\mathbf{e} + \lambda\ \mathrm{tr}(\mathbf{e})\mathbf{1} \tag{13.88}$$

satisfies objectivity and describes an isotropic material. For a tensile test with true stress $T_{11} = \sigma$ and extension $\alpha - 1$,

$$\sigma/E = \ln(\alpha). \tag{13.89}$$

This relation is linear up to three significant figures for $0 \le \sigma/E < 0.001$. The relation is nearly linear even for much larger extensions (Figure 13.5). This constitutive relation is used by ANSYS when the material model is specified as linear elastic isotropic and large displacement effects are enabled (NLGEOM ON).

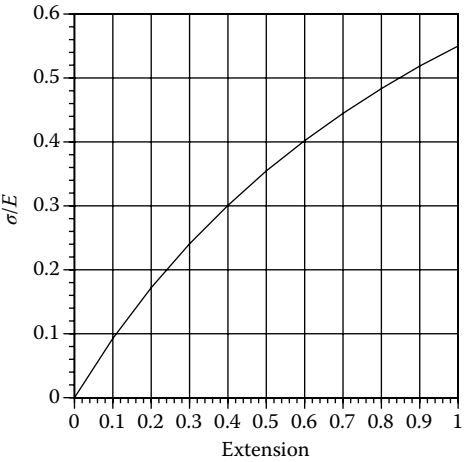


FIGURE 13.5 True stress vs. logarithmic strain.

13.1.9 Yeoh Model

Let us consider the series representation 13.64 but retain only the lower-order terms in I_1:

$$W = a_1(I_1 - 3) + a_2(I_1 - 3)^2 + a_3(I_1 - 3)^3. \tag{13.90}$$

An incompressible model with this strain energy function has been found to provide fit data on carbon black–filled rubber vulcanizates.[7] In this case,

$$\mathbf{T} = -p\mathbf{1} + 2\frac{\partial W}{\partial I_1}\mathbf{B}. \tag{13.91}$$

In a simple shearing motion,

$$\mathbf{B} = \mathbf{1} + k(\mathbf{e}_1\mathbf{e}_2 + \mathbf{e}_2\mathbf{e}_1) + k^2\mathbf{e}_1\mathbf{e}_1. \tag{13.92}$$

The corresponding shear stress is

$$T_{12} = 2a_1k + 4a_2k^3 + 6a_3k^5. \tag{13.93}$$

For small deformations k is the shear strain, so the shear modulus is $\mu = 2a_1$. For filled rubber, the constant a_2 is typically negative and the constant a_3 is a small fraction of the shear modulus.

For the tensile test,

$$\mathbf{B} = \alpha_1^2\mathbf{e}_1\mathbf{e}_1 + \alpha_2^2\mathbf{e}_2\mathbf{e}_2 + \alpha_3^2\mathbf{e}_3\mathbf{e}_3. \tag{13.94}$$

The principal stresses are

$$T_i = -p + 2\alpha_i^2\left(a_1 + 2a_2(I_1 - 3) + 3a_3(I_1 - 3)^2\right). \tag{13.95}$$

Since $\alpha_2 = \alpha_3 = \alpha_1^{-1/2}$ for the incompressible model, we find from $T_2 = 0$ that

$$T_1 = 2(\alpha_1^2 - \alpha_1^{-1})\left(a_1 + 2a_2(I_1 - 3) + 3a_3(I_1 - 3)^2\right). \tag{13.96}$$

The Yeoh model can be extended to compressible materials by introducing the modified invariant $\bar{I}_1$ in place of I_1 and choosing a model for volumetric strain such as

$$W = \sum_n a_n(\bar{I}_1 - 3)^n + \sum_n b_n(J - 1)^{2n}. \tag{13.97}$$

From 13.18,

$$\mathbf{S} = 2a_1 J^{-2/3}\mathbf{1} + \left(2b_1(J^2 - J) - \frac{2}{3}\bar{I}_1 a_1\right)\mathbf{C}^{-1} + \sum_{n=2} 2na_n(\bar{I}_1 - 3)^{n-1} J^{-2/3}\mathbf{1} + \left\{\sum_{n=2}\left(2nb_n(J-1)^{2n-1}J - \frac{2}{3}na_n(\bar{I}_1 - 3)^{n-1}\bar{I}_1\right)\right\}\mathbf{C}^{-1}. \tag{13.98}$$

For small deformations $J \doteq 1 + \text{tr}\,\boldsymbol{\varepsilon}$, $I_1 \doteq 3 + 2\text{tr}\,\boldsymbol{\varepsilon}$, $\bar{I}_1 \doteq 3$, $\mathbf{C}^{-1} \doteq \mathbf{1} - 2\boldsymbol{\varepsilon}$,

$$\mathbf{S} = 4a_1\boldsymbol{\varepsilon} + \left(2b_1 - \frac{2}{3}2a_1\right)\mathbf{1}. \tag{13.99}$$

Therefore, the shear modulus $\mu = 2a_1$ and the bulk modulus $\kappa = 2b_1$.

13.1.10 Fitting Constitutive Relations to Experimental Data

In practice, we begin with the results of material testing and we have to adjust the coefficients in the constitutive model to fit the measured data with adequate accuracy. In principle, the polynomial formula 13.21 can fit any experiment, but in practice we must truncate the series at some point and the determination of the remaining coefficients from the experimental data is difficult. ANSYS provides for direct input of experimental data and determination of the material parameters of the model to fit the data by a linear or nonlinear regression analysis. The minimum experimental test data needed to characterize the material are those from a tensile test and a volumetric deformation.

13.1.10.1 Volumetric Data

The measured data consist of the mean pressure $p = -\text{tr}\,\mathbf{T}/3$ and the ratio J of deformed volume to undeformed volume. For example, for the polynomial model, from 13.73,

$$p = -\sum_{n=1}^{M} n\kappa_n(J-1)^{2n-1}. \tag{13.100}$$

The material constants κ_n are determined by fitting this formula to the measured curve of p versus J.

13.1.10.2 Tensile Test

Suppose the x axis is the loaded direction; for an isotropic material, the coordinate axes are principal axes of stress and strain. The true stress is T_1. The transverse stretches are equal, $\alpha_2 = \alpha_3$, and the ratio of deformed volume to initial volume is $J = \alpha_1\alpha_2^2$. The

nominal stress P_1 is the load divided by the original area, $P_1 = \alpha_2^2 T_1 = J\alpha_1^{-1} T_1$ and the Kirchhoff stress measure is $\hat{T}_1 = JT_1$. One measures α_2, α_1, and P_1. This determines the stress–strain curve $\hat{T}_1$ versus $(\bar{\alpha}_1 - 1)$. For example, for the polynomial model, from 13.70 with $\hat{T}_2 = 0$ and $\bar{\alpha}_2 = \bar{\alpha}_1^{-1/2}$,

$$\begin{aligned}\hat{T}_1 = &\sum_{n=1}^{N} 2c_{n0} n \bar{J}_1^{n-1}\left(\bar{\alpha}_1^2 - \bar{\alpha}_1^{-1}\right) + \sum_{n=1}^{N} 2c_{0n} n \bar{J}_2^{n-1}\left(\bar{\alpha}_1 - \bar{\alpha}_1^{-2}\right) \\ &+ \sum_{n=1}^{N}\sum_{m=1}^{N} 2c_{nm}\left(n\bar{J}_1^{n-1}\bar{J}_2^{m}\left(\bar{\alpha}_1^2 - \bar{\alpha}_1^{-1}\right) + m\bar{J}_1^{n}\bar{J}_2^{m-1}\left(\bar{\alpha}_1 - \bar{\alpha}_1^{-2}\right)\right).\end{aligned} \tag{13.101}$$

A curve fit to the tensile test data determines the material constants c_{nm}.

13.1.10.3 Biaxial Test

A thin sheet is loaded by tension in two orthogonal directions in the plane of the sheet: $T_1 = T_2$, $T_3 = 0$, $\alpha_2 = \alpha_1$. From 13.70–13.72 with $\bar{\alpha}_1^2 \bar{\alpha}_3 = 1$,

$$\hat{T}_1 = 2c_{10}\left(\bar{\alpha}_1^2 - \bar{\alpha}_1^{-4}\right) + 2c_{01}\left(\bar{\alpha}_1^{-4} - \bar{\alpha}_1^{-2}\right). \tag{13.102}$$

This test can be used in place of the tensile test or in addition to it.

Examples are provided in Sections 15.37 and 15.38. If no volume data are entered, ANSYS assumes an incompressible material (except for the Blatz–Ko model). The ANSYS program sometimes returns unrealistic values and must be used only as a starting point and with caution.

13.2 FINITE ELEMENTS FOR LARGE DISPLACEMENTS

The equilibrium condition is expressed by the virtual work formula 13.10:

$$\int_{\mathcal{R}} P_{ij} \frac{\partial \phi_i}{\partial X_j} \, \mathrm{d}V = \int_{\partial\mathcal{R}} p_i \phi_i \, \mathrm{d}A + \int_{\mathcal{R}} \rho_0 b_i \phi_i \, \mathrm{d}V. \tag{13.103}$$

Integration is over the chosen reference configuration. In this section, the reference configuration is the unstressed initial body. We divide that configuration into finite elements and generate the variational function ϕ by using the shape functions for the elements:

$$\phi_i = \sum_K N_{iK} \bar{D}_K \left\{\phi_i\right\} = \left[N_{iK}\right]\left\{\bar{D}_K\right\}, \quad \boldsymbol{\phi} = \mathbf{N}\bar{\mathbf{D}}, \tag{13.104}$$

where $\bar{\mathbf{D}}$ denotes arbitrary (virtual) nodal displacements. The gradients of $\boldsymbol{\phi}$ are

$$\frac{\partial \phi_i}{\partial X_j} = \sum_K A_{ijK} \bar{D}_K, \quad A_{ijK} = \frac{\partial N_{iK}}{\partial X_j}, \tag{13.105}$$

or, arranging the gradient in a column matrix using the FORTRAN ordering,

$$\left\{ \frac{\partial \phi_m}{\partial X_k} \right\} = \begin{Bmatrix} \phi_{1,1} \\ \phi_{2,1} \\ \phi_{3,1} \\ \cdot \\ \cdot \\ \cdot \end{Bmatrix} = \left[A_{\alpha K} \right] \left\{ \bar{D}_K \right\} = \mathbf{A}\bar{\mathbf{D}}, \quad A_{\alpha K} = A_{ijK}, \quad \alpha = i + 3(j-1). \tag{13.106}$$

Since the virtual work formula 13.103 must hold for arbitrary $\bar{D}_K$, we find the following finite element version of the virtual work formula in matrix notation:

$$\sum_m \int_{\mathcal{R}_m} \mathbf{A}^{\mathrm{T}} \mathbf{P} \, \mathrm{d}V = \sum_m \int_{\partial \mathcal{R}_m} \mathbf{N}^{\mathrm{T}} \mathbf{p} \, \mathrm{d}A + \sum_m \int_{\partial \mathcal{R}_m} \mathbf{N}^{\mathrm{T}} \mathbf{b} \rho_0 \, \mathrm{d}V, \tag{13.107}$$

where the components of the Piola tensor **P**, the surface traction **p** per unit undeformed area, and the body force **b** per unit mass, are put into column matrices using the FORTRAN ordering: $\mathbf{P} = \{P_\alpha\}$, $P_\alpha = P_{ij}$, $\alpha = i + 3(j - 1)$.

The right-hand side of 13.107 is the usual matrix of given external nodal **F** forces at any time t. The left-hand side is the matrix of internal nodal forces **f** due to stresses that balance the external forces. Equation 13.107 therefore simply states that

$$\mathbf{f}(t) = \mathbf{F}(t), \tag{13.108}$$

where

$$\begin{aligned} \mathbf{f}(t) &= \sum_m \int_{\mathcal{R}_m} \mathbf{A}^{\mathrm{T}} \mathbf{P}(t) \, \mathrm{d}V, \\ f_K &= \sum_m \int_{\mathcal{R}_m} A_{ijK} P(t)_{ij} \, dV. \end{aligned} \tag{13.109}$$

The basic equations for large displacements are nonlinear. In order to solve them, one must resort to step-by-step or to iterative procedures. For each step in time, we have

$$\mathrm{d}\mathbf{f} = \sum_m \int_{\mathcal{R}_m} \mathbf{A}^{\mathrm{T}} \, \mathrm{d}\mathbf{P} \, \mathrm{d}V. \tag{13.110}$$

The reference configuration is arbitrary but the Piola tensor depends on the choice of reference configuration.

13.2.1 Lagrangian Formulation

Let us first suppose that the reference configuration is the initial configuration of the body, which is also the unstressed configuration. The reference configuration is fixed throughout the calculation. Whatever the actual form of the constitutive relation of elasticity, we see from 13.14 and 13.15, that **P** is a nonlinear function of **G**:

$$\mathbf{P} = \mathcal{F}(\mathbf{G}). \tag{13.111}$$

Hence,

$$\begin{aligned} \mathrm{d}\mathbf{P} &= \mathbf{C}_{\mathrm{T}}\mathrm{d}\mathbf{G}, \\ \mathrm{d}P_{ij} &= C^{\mathrm{T}}_{ijkm}\mathrm{d}G_{km}, \end{aligned} \tag{13.112}$$

where $\mathbf{C}_{\mathrm{T}}$ is the matrix of derivatives of $\mathcal{F}$ with respect to **G** that we will call the tangent modulus matrix: $\mathbf{C}_{\mathrm{T}} = \mathbf{C}_{\mathrm{T}}(\mathbf{G})$. Thus, 13.110 becomes

$$\mathrm{d}\mathbf{f} = \sum_m \int_{\mathcal{R}_m} \left(\mathbf{A}^{\mathrm{T}}\mathbf{C}_{\mathrm{T}}\mathrm{d}\mathbf{G}\right)\mathrm{d}V. \tag{13.113}$$

From 13.4, if d**x** is the incremental change in position vector,

$$\mathrm{d}G_{km} = \frac{\partial}{\partial X_m}\mathrm{d}x_k. \tag{13.114}$$

We can use the finite element shape functions to approximate the incremental position vector d**x** in each element

$$\begin{aligned} \mathrm{d}\mathbf{x} &= \mathbf{N}\mathrm{d}\mathbf{D}, \\ \mathrm{d}x_k &= N_{kM}\mathrm{d}D_M, \end{aligned} \tag{13.115}$$

where d**D** is the incremental change in nodal displacements for a step change in time. Note that **N** depends only on **X** and not on time t since the reference configuration is fixed. Then,

$$\mathrm{d}\mathbf{G} = \mathbf{A}\mathrm{d}\mathbf{D} \tag{13.116}$$

for each element, and formula 13.113 for change in internal nodal forces becomes

$$\mathrm{d}\mathbf{f} = \mathbf{K}_{\mathrm{T}}\mathrm{d}\mathbf{D}, \tag{13.117}$$

where

$$\mathbf{K}_{\mathrm{T}} = \sum_{m} \int_{\mathcal{R}_m} (\mathbf{A}^{\mathrm{T}} \mathbf{C}_{\mathrm{T}} \mathbf{A})\, \mathrm{d}V,$$
$$K_{\mathrm{RS}}^{\mathrm{T}} = \sum_{m} \int_{\mathcal{R}_m} A_{ijR} C_{ijkm}^{\mathrm{T}} A_{kmS}\, \mathrm{d}V, \tag{13.118}$$

is the tangent stiffness matrix formed by merging the tangent stiffness matrix for each element. We will prove below that $\mathbf{C}_{\mathrm{T}}$ is symmetric ($C_{ijkm}^{\mathrm{T}} = C_{kmij}^{\mathrm{T}}$) so that $\mathbf{K}_{\mathrm{T}}$ is symmetric.

13.2.2 Basic Step-by-Step Analysis

The solution can now be constructed via step-by-step or iteration procedures. For the basic step-by-step procedure,

$$\mathbf{f}(t + \mathrm{d}t) = \mathbf{f}(t) + \mathrm{d}\mathbf{f}(t) \tag{13.119}$$

and we seek the incremental nodal displacement such that

$$\mathbf{f}(t + \mathrm{d}t) = \mathbf{F}(t + \mathrm{d}t). \tag{13.120}$$

Using 13.117,

$$\mathbf{K}_{\mathrm{T}}(t)\mathrm{d}\mathbf{D}(t) = \mathbf{F}(t + \mathrm{d}t) - \mathbf{f}(t). \tag{13.121}$$

F is given and *f* is determined by 13.109. Given the solution up to time *t*, we calculate the incremental displacement d**D**(*t*) and march the solution forward in the usual manner:

$$\mathbf{D}(t + \mathrm{d}t) = \mathbf{D}(t) + \mathrm{d}\mathbf{D}(t). \tag{13.122}$$

This procedure requires a load correction in each step. It is sometimes sufficiently accurate to set $\mathbf{F}(t + \mathrm{d}t) = \mathbf{F}(t) + \mathrm{d}\mathbf{F}(t)$ and $\mathbf{f}(t) = \mathbf{F}(t)$. Then, the equations to be formed and solved simplify to

$$\mathbf{K}_{\mathrm{T}}(t)\mathrm{d}\mathbf{D}(t) = \mathrm{d}\mathbf{F}(t) \tag{13.123}$$

for each step. The steps will, of course, have to be smaller than if 13.121 is used.

13.2.3 Iteration Procedure

As the basic step-by-step procedure proceeds, there is an increasing error that produces an imbalance between the internal and external node forces. We can use the

fundamental relation 13.117 to determine corrections to the node displacements, and therefore to the stresses, which will correct the unbalanced nodal forces. At any fixed load **F**, having some approximation $\mathbf{f}^i$ for the internal node forces, we seek corrections such that

$$\mathbf{f}^i + \mathrm{d}\mathbf{f}^i = \mathbf{F}. \tag{13.124}$$

Using 13.117, we have

$$\mathbf{K}_{\mathrm{T}}^i \mathrm{d}\mathbf{D}^i = \mathbf{R}^i \equiv \mathbf{F} - \mathbf{f}^i \tag{13.125}$$

where tangent stiffness is constructed from the existing approximation for stress and deformation. It can be updated after each iteration or not. From the calculated improvement $\mathrm{d}\mathbf{D}^i$ in nodal displacement, we calculate the improvement in displacement gradient by 13.116, in stress from 13.111, and in internal node forces **f** by 13.110. This process is repeated until the unbalanced node forces $\mathbf{R}^i = \mathbf{F} - \mathbf{f}^i$ are sufficiently small.

13.2.4 Updated Reference Configuration

The procedure described above, where the calculations are referred to as a single unstressed reference configuration, is called the *Lagrangian* procedure, or the total Lagrangian procedure, in the finite element literature. Note that the external tractions are the loads per unit area of the reference configuration.

It is also possible to change the reference configuration before any step in the calculation. If it is changed to the current configuration, the procedure is known as the *updated* Lagrangian method. If the loads are specified relative to the deformed configuration, the updated Lagrangian method can handle the surface tractions directly, whereas the Lagrangian formulation cannot.

The Piola tensor 13.8 depends on the reference configuration. If the reference configuration is changed in the virtual work formulation 13.10, it is necessary to introduce the Piola tensor with respect to that new reference configuration. Let $\widehat{X}_k$ denote the coordinates of a material particle in the updated configuration (Table 13.3).

Define the respective deformation gradients by

$$G_{km} = \frac{\partial x_k}{\partial X_m}, \quad G^{\mathrm{o}}_{km} = \frac{\partial \widehat{X}_k}{\partial X_m}, \quad \widehat{G}_{km} = \frac{\partial x_k}{\partial \widehat{X}_k}. \tag{13.126}$$

TABLE 13.3
Reference Configurations

Position	Density
X	ρ_0
$\widehat{\mathbf{X}}$	$\bar{\rho}$
X	ρ

Note that

$$G_{km} = \widehat{G}_{ki} G^{\circ}_{im}. \tag{13.127}$$

The Piola tensor with respect to the updated configuration is

$$\widehat{P}_{jm} = \frac{\widehat{\rho}}{\rho} \widehat{G}^{-1}_{mi} T_{ij}. \tag{13.128}$$

Using 13.8, we find that

$$\widehat{P}_{jm} = \frac{\widehat{\rho}}{\rho_0} G^{\circ}_{mk} P_{jk}. \tag{13.129}$$

For the step-by-step or iteration analysis, we also need the relation of the increment of $\widehat{\mathbf{P}}$ to the increment of $\widehat{\mathbf{G}}$:

$$\mathrm{d}\widehat{P}_{jm} = \widehat{c}_{mjri} \mathrm{d}\widehat{G}_{ri}. \tag{13.130}$$

From 13.112, 13.127, and 13.129, we find

$$\mathrm{d}\widehat{P}_{jm} = \frac{\widehat{\rho}}{\rho_0} G^{\circ}_{mk} G^{\circ}_{is} c_{kjrs} \mathrm{d}\widehat{G}_{ri}. \tag{13.131}$$

Thus,

$$\widehat{c}_{mjri} = \frac{\widehat{\rho}}{\rho_0} G^{\circ}_{mk} G^{\circ}_{is} c_{kjrs}. \tag{13.132}$$

With respect to the updated configuration $\widehat{\mathcal{R}}$ the internal resisting nodal forces are

$$\mathbf{f}(t) = \sum_m \int_{\widehat{\mathcal{R}}_m} \widehat{\mathbf{A}}^{\mathrm{T}} \widehat{\mathbf{P}} \, \mathrm{d}V \tag{13.133}$$

in place of 13.109. Using 13.130, we have

$$\mathrm{d}\mathbf{f} = \widehat{\mathbf{K}}_{\mathrm{T}} \mathrm{d}\mathbf{D} \tag{13.134}$$

in place of 13.117, with

$$\widehat{\mathbf{K}}_{\mathrm{T}} = \sum_m \int_{\widehat{\mathcal{R}}_{\mathrm{m}}} (\widehat{\mathbf{A}}^{\mathrm{T}} \widehat{\mathbf{C}}_{\mathrm{T}} \widehat{\mathbf{A}}) \, \mathrm{d}V. \tag{13.135}$$

The element matrices $\widehat{\mathbf{A}}$ can be determined from an entirely new element layout for each configuration, but in the updated Lagrangian procedure one typically generates the new element configuration by using the original element layout with updated nodal coordinates.

The external nodal forces with respect to the updated configuration are

$$\mathbf{F} = \sum_m \int_{\partial \mathcal{R}_m} \widehat{\mathbf{N}}^{\mathrm{T}} \widehat{\mathbf{p}} \, \mathrm{d}A + \sum_m \int_{\mathcal{R}_m} \widehat{\mathbf{N}}^{\mathrm{T}} \widehat{\mathbf{b}} \widehat{\rho} \, \mathrm{d}V, \tag{13.136}$$

and $\widehat{\mathbf{p}}$ is the traction per unit area of the configuration $\widehat{\mathbf{R}}$.

The solution now proceeds using the step-by-step formula or the iteration formula. Note, however, that the solution to elasticity problems with large displacements may not be unique and a single-step solution may locate an unexpected equilibrium configuration. Thus, it is generally best to trace the solution step by step and use iteration within the step to refine the solution. This is especially true if buckling is possible in any situation.

13.2.5 Example I

Table 13.4 shows the analysis of the tensile test using ANSYS for the incompressible Mooney–Rivlin material for the case $c_1 = c_2$. The "load" is P_{11}/c_1 and the stretch is α_1. This is a 3D analysis using the element Solid185 for which one-element models the tensile test exactly. The total load was applied in 10 steps with program-chosen iterations in each step using the default convergence criterion (L2 norm on force with a tolerance of 0.001). This resulted in two iterations in step 1 and increased to 20 iterations in step 10. The results agree with the exact formula 13.52 to five decimal places.

13.2.6 Example II

As a second example, let us consider a sheet of rubber that is bonded to a rigid support along one side and extended by a uniform tension along the opposite side.

TABLE 13.4
Tensile Test of Mooney–Rivlin Material

P_{11}/c_1 by FEA	α_1	P_{11}/c_1 by (28.47)
0	1	0
0.4	1.03508	0.40000
0.8	1.07403	0.80000
1.2	1.11746	1.20000
1.6	1.16609	1.60000
2	1.22074	2.00000
2.4	1.28235	2.40000
2.8	1.35190	2.80000
3.2	1.43041	3.20000
3.6	1.51887	3.60000
4.0	1.61803	4.00000

We can treat the sheet as a plane stress problem. The rubber was analyzed as a Mooney–Rivlin Material with $c_1 = c_2 = 0.5$ psi and $\kappa^{-1} = 0.001(\nu = 0.499)$. The sheet is a 1-unit square and it is divided into 400 equal square elements that are formed from four-node Solid 182 with reduced integration. The total load of $P_{11} = 2.5$ psi was applied in 10 steps with program chosen iterations in each step using the default convergence criterion (L2 norm on force with a tolerance of 0.001). This resulted in two iterations in each step. The resulting extension is $U_x = 0.873634$. The calculated load displacement is shown in Figure 13.6. Figure 13.7 shows the deformed shape. Note the wavy edge calculated with this rather coarse mesh.

The procedure based on 13.125 is known as the Newton–Raphson procedure. Modifications in this procedure abound. For instance, one may use for $\mathbf{K}_T$ the stiffness matrix for some prior iteration, even the initial matrix for zero load. One may also update the stiffness matrix only after several iterations, and so forth.

This iteration for a given load can be applied after each step of the step-by-step method in order to improve the accuracy of that step. In the extreme case, it can be applied to reach the solution for the total load in one step, since the elasticity solution is not history- or path-dependent.

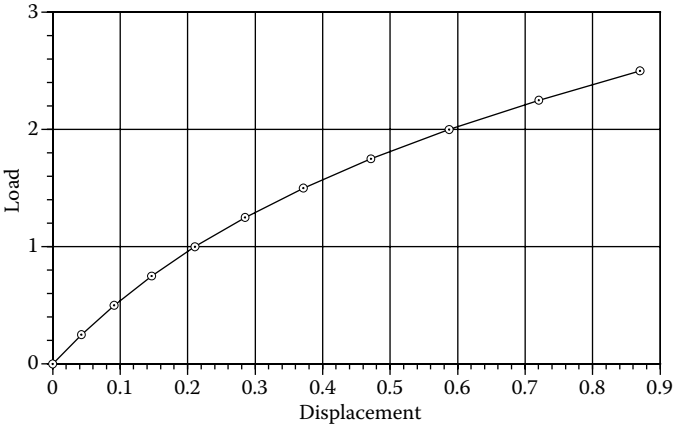


FIGURE 13.6 Stretching of a Mooney–Rivlin sheet.

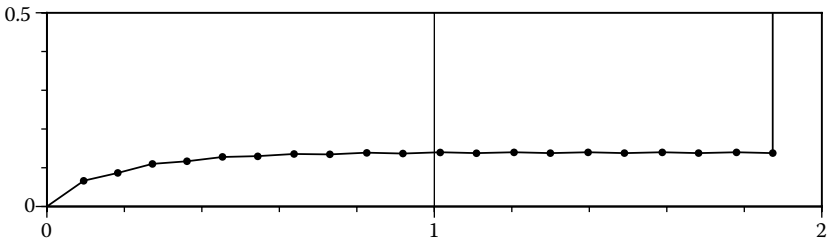


FIGURE 13.7 Deformed shape in sheet tension of a Mooney–Rivlin material.

Most frequently, one uses the step-by-step procedure to make the initial approximation, and then iteration to improve that approximation, before marching forward another step. This allows one to take rather large steps and maintain an acceptable magnitude of error in the solution.

13.3 STRUCTURE OF TANGENT MODULUS

For an elastic material, the dependence of **P** on **G** is restricted by the requirement that **S** is derived by the gradient of the strain energy. From 13.8 and 13.12,

$$P_{mk} = S_{ki} G_{mi}. \tag{13.137}$$

Therefore,

$$\dot{P}_{mk} = \dot{S}_{ki} G_{mi} + S_{ki} \dot{G}_{mi}. \tag{13.138}$$

From 13.15 and 13.17,

$$\dot{S}_{ij} = 2 \frac{\partial^2 W}{\partial C_{ij} \partial C_{km}} \dot{C}_{km} = \overline{c}_{ijkm} \dot{C}_{km} = 2 \overline{c}_{ijks} G_{rk} \dot{G}_{rs}. \tag{13.139}$$

Therefore,

$$\overline{c}_{ijkm} = \overline{c}_{jikm} = \overline{c}_{ijmk} = \overline{c}_{kmij}. \tag{13.140}$$

Therefore,

$$\dot{P}_{mk} = 2 \overline{c}_{kips} G_{rp} G_{mi} \dot{G}_{rs} + S_{ks} \delta_{mr} \dot{G}_{rs}. \tag{13.141}$$

The tangent modulus in relation 13.112 has the form

$$\dot{P}_{km} = c^{\mathrm{T}}_{kmrs} \dot{G}_{rs}. \tag{13.142}$$

The tangent modulus therefore has the form

$$c^{\mathrm{T}}_{kmrs} = c^{0}_{kmrs} + c^{1}_{kmrs}, \tag{13.143}$$

where

$$c^{0}_{mkrs} = 2 \overline{c}_{kips} G_{rp} G_{mi}, \tag{13.144}$$

$$c^{1}_{mkrs} = S_{ks} \delta_{mr}. \tag{13.145}$$

We see that

$$c^{\mathrm{T}}_{kmrs} = c^{\mathrm{T}}_{rskm}, \tag{13.146}$$

which proves that the tangent modulus is symmetric.

Substituting 13.143 into 13.118, one finds that the stiffness matrix has the form

$$\mathbf{K}_T = \mathbf{K}_0 + \mathbf{K}_1, \tag{13.147}$$

with an obvious notation. The part $\mathbf{K}_0$ of the tangent stiffness matrix is similar to the stiffness of the linear theory but corrected for large displacements. The matrix $\mathbf{K}_1$ is the correction for the direction of components of the internal stress in balancing the nodal forces. If the internal stresses are tension, this produces a stress stiffening. If the internal stresses are compressive, this correction can actually add to the effective external loads to produce additional displacements, and if the current stresses are large enough negatively so that

$$\det(\mathbf{K}_0 + \mathbf{K}_1) = 0, \tag{13.148}$$

and buckling occurs. The solution can then be stepped further only when inertial forces are included.

The significance of the terms $\mathbf{K}_0$ and $\mathbf{K}_1$ is more apparent if the current configuration is chosen as the reference configuration for the next step in the calculations. In this case, $\widehat{G}_{km} = \delta_{km}$ and $G^0_{km} = G_{km}$. From 13.132, 13.143, and 13.145:

$$\widehat{c}_{ijrk} = \frac{\widehat{\rho}}{\rho_0} G_{im} G_{ks} c^0_{mjrs} + \frac{\widehat{\rho}}{\rho_0} G_{im} G_{ks} S_{ms} \delta_{jr}. \tag{13.149}$$

By 13.14, the last term depends only on the current stress*:

$$\widehat{c}_{ijrk} = \frac{\widehat{\rho}}{\rho_0} G_{im} G_{ks} c^0_{mjrs} + \frac{\widehat{\rho}}{\rho_0} T_{ik} \delta_{jr}. \tag{13.150}$$

That is, $\mathbf{K}_1$ depends on the current stress.

Buckling problems often involve small displacements before buckling occurs. For small deformations, $G_{ij} \approx \delta_{ij}$ and $\rho \approx \rho_0$, and Equation 13.150 reduces to

$$\widehat{c}_{ijrk} = c_{ijrk} + T_{ik} \delta_{jr}, \tag{13.151}$$

where the first term represents the elasticity coefficients for small strains. This leads to formulation of buckling as an eigenvalue problem.†

13.4 STABILITY AND BUCKLING

In the nonlinear theory, some interesting features can occur that do not occur in the linear theory. In particular, all of the various kinds of buckling phenomena (bifurcation or snap-through) may be encountered.

* ANSYS activates the first term by NLGEOM ON and the second term by SSTIFF ON.

† ANSYS adds the second term by PRESTRESS ON.

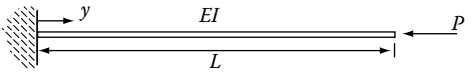


FIGURE 13.8 Axially loaded column.

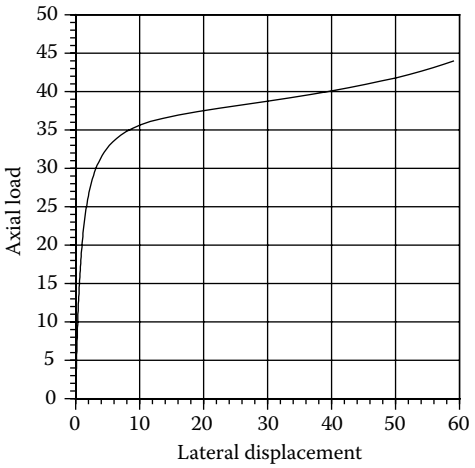


FIGURE 13.9 Post-buckling of an imperfect column.

13.4.1 Beam–Column

A column with one end fixed and the other free has an axial load P as shown in Figure 13.8. The cross section is 0.5 in.^{2}, $A = 0.25$, $I = 1/192$, $L = 100$ in., $E = 30 \times 10^6$ lb/in.^{2}. The Euler buckling load is $P_{cr} = 0.25\pi^2 EI/L^2 = 38.553$ lb. The column is modeled by 10 Beam3 elements of equal length. The buckling load by the FEM is 38.553 (see Section 15.23). In a real situation, the system is not perfect and lateral displacement of the column will occur. As the buckling load is approached the lateral displacement becomes large, and a large deformation formulation is needed to investigate this phenomenon. A small lateral load can be applied to simulate an imperfection. Figure 13.9 shows the lateral displacement if the load is ramped up to $P = 44$ lb, exceeding the Euler buckling load (see Section 15.24).

13.5 SNAP-THROUGH BUCKLING

If det $\mathbf{K}_T = 0$ or if there is no nearby equilibrium configuration for the next step in external load, the basic methods described above will fail. This can happen for structures that exhibit the oil-canning or snap-through behavior (Figure 13.10). To extend the solution beyond the critical point, one needs to allow the external loads to decrease in the next step.

To formulate such a procedure, let us consider the so-called proportional loading: $\mathbf{F}(t) = \lambda(t)\mathbf{F}_m$, where λ is a scalar multiplier and $\mathbf{F}_m$ is a matrix of reference loads,

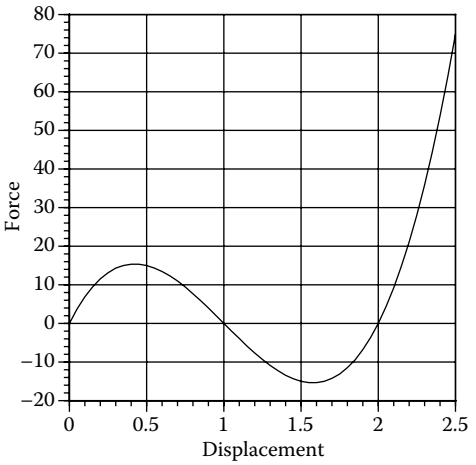


FIGURE 13.10 Snap-through buckling.

typically the maximum loads to be applied in the analysis. The balance of internal and external node forces becomes

$$\lambda(t)\mathbf{F}_{\mathrm{m}} = \mathbf{f}(t). \tag{13.152}$$

For a step Δt, we have

$$\begin{aligned} \mathbf{F}(t+\Delta t) &= \mathbf{f}(t+\Delta t), \\ (\lambda(t)+\Delta\lambda)\mathbf{F}_{\mathrm{m}} &= \mathbf{f}(t) + \mathbf{K}_{\mathrm{T}}(t)\Delta\mathbf{D}, \\ \mathbf{K}_{\mathrm{T}}(t)\Delta\mathbf{D} &= (\lambda(t)+\Delta\lambda)\mathbf{F}_{\mathrm{m}} - \mathbf{f}(t). \end{aligned} \tag{13.153}$$

We now suppose that a solution, λ_{a} and $\mathbf{D}_{\mathrm{a}}$, has been established for some loading, and seek an extension of the solution to a nearby configuration by the iteration scheme

$$\mathbf{K}_{\mathrm{T}}\Delta\mathbf{D}^{i} = (\lambda^{i} + \Delta\lambda^{i})\mathbf{F}_{\mathrm{m}} - \mathbf{f}^{i}. \tag{13.154}$$

The solution of this relation has the form

$$\Delta D^{i} = \Delta\lambda^{i}\mathbf{D}_{\mathrm{m}} + \Delta\bar{\mathbf{D}}^{i}, \tag{13.155}$$

where

$$\mathbf{K}_{\mathrm{T}}\mathbf{D}_{\mathrm{m}} = \mathbf{F}_{\mathrm{m}}, \tag{13.156}$$

$$\mathbf{K}_{\mathrm{T}}\Delta\bar{\mathbf{D}}^{i} = \lambda^{i}\mathbf{F}_{\mathrm{m}} - \mathbf{f}^{i}. \tag{13.157}$$

We need a method for determination of $\Delta\lambda^i$. The arc length method[8–10] can be used.

A step change in $\mathbf{D}$ and λ has length s given by (using nondimensional variables)

$$s^2 = (\lambda^i - \lambda_a)^2 + \sum_K (D_K^i - D_K^a)^2 . \tag{13.158}$$

This is illustrated in Figure 13.11 for a single degree of freedom system. We choose the first step so that s is small and hold s constant for subsequent iterations. This allows the load parameter λ to decrease if necessary while the magnitude of displacement increases. The derivative of 13.158 provides the constraint relation:

$$(\lambda^i - \lambda_a)\Delta\lambda^i + \sum_K (D_K^i - D_K^a)\Delta D_K^i = 0. \tag{13.159}$$

Using 13.155, we find

$$\Delta\lambda^i = \frac{-\sum_K (D_K^i - D_K^a)\Delta\bar{D}_K^i}{\lambda^i - \lambda_a + \sum_K (D_K^i - D_K^a)D_K^m} . \tag{13.160}$$

The increment in displacement can now be calculated from 13.155. The tangent stiffness can be unchanged in each iteration or updated as corrections are made to the updated reference configuration.

The finite steps calculated by 13.159 will result in small increases in arc length s defined by 13.158 as shown in Figure 13.11. For a single degree of freedom, let

$$\mathbf{t}^i = (D^i - D^a)\mathbf{e}_1 + (\lambda^i - \lambda^a)\mathbf{e}_2, \tag{13.161}$$

$$\mathbf{n}^i = \Delta D^i \mathbf{e}_1 + \Delta\lambda^i \mathbf{e}_2. \tag{13.162}$$

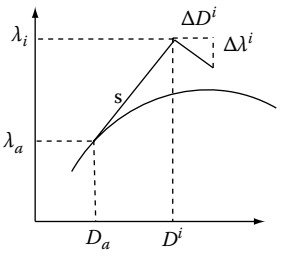


FIGURE 13.11 Arc length method.

The condition $|\mathbf{t}^i| = s$ and condition 13.159 are equivalent to the condition $\mathbf{t}^i \cdot \mathbf{n}^i = 0$, that $\mathbf{t}^i$ and $\mathbf{n}^i$ are orthogonal. Each iteration to $\mathbf{t}^{i+1} = \mathbf{t}^i + \mathbf{n}^i$ will therefore increase the arc length slightly to $a = |\mathbf{t}^{i+1}|$:

$$a^2 = s^2 + (\Delta\lambda^i)^2 + (\Delta D^i)^2. \tag{13.163}$$

A simple correction can be made to preserve the arc length.[11] The vector

$$\mathbf{r}^i = -\frac{a-s}{a}\mathbf{t}^{i+1} \tag{13.164}$$

is the correction needed to preserve the arc length. From the geometry of the right triangle in Figure 13.11, we find

$$\mathbf{t}^i \cdot \mathbf{r}^i = -\frac{a-s}{a}\mathbf{t}^i \cdot \mathbf{t}^{i+1} = -\frac{(a-s)s^2}{a}. \tag{13.165}$$

The corrected step $\mathbf{m}^i = \mathbf{n}^i + \mathbf{r}^i$ is not orthogonal to $\mathbf{t}^i$. Let

$$R^i = \mathbf{t}^i \cdot \mathbf{m}^i = \mathbf{t}^i \cdot \mathbf{r}^i = -\frac{(a-s)s^2}{a}. \tag{13.166}$$

The corrected increments $\Delta\lambda^i$ and ΔD^i are given by

$$\mathbf{m}^i = \Delta D^i \mathbf{e}_1 + \Delta\lambda^i \mathbf{e}_2. \tag{13.167}$$

Using 13.155 and inserting into 13.166, we find the formula for the corrected load step:

$$\Delta\lambda^i = \frac{R^i - (D^i - D_a)\Delta\bar{D}^i}{\lambda^i - \lambda_a + (D^i - D_a)D_m}. \tag{13.168}$$

The modified procedure for multiple degrees of freedom is as follows:

1. $\Delta\lambda^i = \dfrac{-\sum_K (D_K^i - D_K^a)\Delta\bar{D}_K^i}{\lambda^i - \lambda_a + \sum_K (D_K^i - D_K^a)D_K^m}$ (causes growth in arc length)
2. $\Delta\mathbf{D}^i = \Delta\lambda^i \mathbf{D}_m + \Delta\bar{\mathbf{D}}^i$
3. $a^2 = s^2 + (\Delta\lambda^i)^2 + \left|\Delta D^i\right|^2$ (increased arc length)
4. $R^i = -\dfrac{(a-s)s^2}{a}$ (correction factor)
5. $\Delta\lambda^i = \dfrac{R^i - \sum_K (D_K^i - D_K^a)\Delta\bar{D}_K^i}{\lambda^i - \lambda_a + \sum_K (D_K^i - D_K^a)D_K^m}$ (correction to load factor)
6. $\Delta D^i = \Delta\lambda^i \mathbf{D}_m + \Delta\bar{\mathbf{D}}^i$ (using corrected load factor)

13.5.1 Shallow Arch

The procedure will be illustrated by application to a shallow arch formed by two axial-force members (Figure 13.12).

Each member has an area A, a modulus E, and an axial tensile force P. The only degree of freedom is the downward displacement D of the joint. Because of the symmetry, we only need to consider the left half of the structure. The internal nodal resisting force is

$$f = P\frac{(V-D)}{L}. \tag{13.169}$$

The constitutive relation for the member is

$$P = P(L), \quad \mathrm{d}P = C\,\mathrm{d}L, \quad C = \frac{\partial P}{\partial L}. \tag{13.170}$$

The deformed length L is related to the displacement by

$$L^2 = H^2 + (V-D)^2, \tag{13.171}$$

so that

$$\mathrm{d}L = -\frac{V-D}{L}\mathrm{d}D. \tag{13.172}$$

Therefore,

$$\mathrm{d}F = K\mathrm{d}D, \tag{13.173}$$

where

$$K = C\frac{(V-D)^2}{L^2} + \frac{P}{L}\left(1 - \frac{(V-D)^2}{L^2}\right). \tag{13.174}$$

If P is a sufficiently large negative (compression) number, the stiffness goes to zero and the arch buckles.

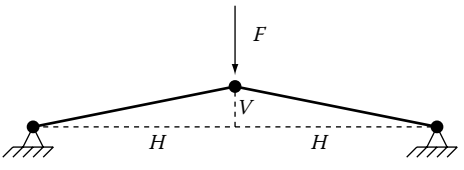


FIGURE 13.12 Shallow arch.

If the rise V is small compared to the length, the extension of the rod is small so the specific constitution relation used is not significant. The simplest example is

$$P = \frac{AE}{L_0}(L - L_0), \tag{13.175}$$

where A is the area, E is the modulus of elasticity, and L_0 is the initial length. Numerical calculations were made for $H = 50$, $V = 1$, and $AE = 10^7$. The steps in the calculation are listed in Section 15.25. The calculated and exact relations agree and are as shown in Figure 13.10. Of course, the portion of the curve with negative slope is statically unstable and if the load is applied as a ramp function, a jump to the right branch will occur dynamically and inertial forces must be included if one is to track the actual deformations.

The maximum and minimum points on the graph are points where $dF = 0$ and therefore $K = 0$. From 13.174, using 13.171 and 13.175 to eliminate P and $(V - D)$, we find $L/H = (L_0/H)^{1/3}$. Then, from 13.171,

$$D = V \mp H\sqrt{(L/H)^2 - 1}, \tag{13.176}$$

from which the corresponding value of F can be calculated.

13.6 PROBLEMS

1. For the tensile test, $x_1 = \alpha_1 X_1$, $x_2 = \alpha_2 X_2$, $x_3 = \alpha_3 X_3$. If the total axial load is F and T_{11} is the true axial stress, prove that $P_{11} = \alpha_2\alpha_3 T_{11}$ and $S_{11} = \dfrac{P_{11}}{\alpha_1}$.
2. Determine the relation between the axial load and the extension of the rod in a tensile test if the material is a Mooney–Rivlin material with $c_1 = c_2$.
3. Consider the tensile test of a Blatz–Ko foam.
   (a) Show that $\dfrac{1}{\mu}P_{11} = \alpha_1^{-1/2} - \alpha_1^{-3}$ and $\dfrac{1}{\mu}T_{11} = 1 - \alpha_1^{-5/2}$.
   (b) Given the initial length = 1 and $P_{11} = 40$, calculate T_{11}. Determine the displacement of the other end if the end $X_1 = 0$ is fixed.
4. Given: a Blatz–Ko material with $\mu = 100$. Consider a tensile test of a cube of side length 1 unit, subjected to a force of 40 units normal to the edges $x =$ constant.
   (a) Use ANSYS to determine the stretches and the true stress (see Section 15.22). Note that you only need one element since the displacements are uniform over the element. Use the Solid 185 element. Use t =1 and a step size of 0.1.
   (b) Compare the solution by ANSYS to the exact formula.
5. Consider stretching of a nonlinear elastic sheet (Figure 13.13) with $a = b = 1$, $p = 100$, a Blatz–Ko material ($\mu = 200$). Use a four-node plane stress element (Plane182). Use the upper half of the sheet and symmetry conditions.

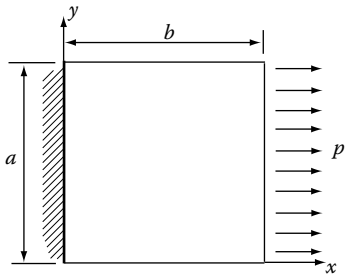


FIGURE 13.13 Stretching of a sheet.

Note that $p = 100$ is a given load per unit reference area and not a pressure per unit deformed area. Submit:

(a) A list of load p versus $u(0,1)$

(b) A graph of load p versus $u(0,1)$ (see Section 15.21)

6. Given a steel column ($E = 30 \times 10^6$ psi, $\nu = 0.3$), of length 10 in., with a square cross section 1×1 in. ($A = 1$, $I = 1/12$). One end is fixed and the other is free (Figure 13.14).
   (a) Determine the buckling load by ANSYS using 10 Beam3 elements (Section 15.23).
   (b) Compare that result with the theoretical buckling load.
7. Using ANSYS, determine the buckling load for the column in problem 6 if it is a pin-ended column.
8. Consider an axially loaded column with one end fixed and the other free. The cross section is 0.5 in.^{2} and the length is 100 in. The material is steel with $E = 30 \times 10^6$ psi and Poisson ratio of 0.3.
   (a) Use ANSYS to determine the Euler buckling load (Answer: $P_{cr} = 38.553$ lb).
   (b) Use ANSYS to determine the post buckled shape for loading of $P = 44$ lb. Apply a small lateral force to instigate lateral displacement. (See Section 15.24.)
9. Use ANSYS to determine the equilibrium configurations for the linkage of Figure 13.12 with $H = 50$, $V = 1$, $F = 75$. For each member, $A = 1$, $E = 10 \times 10^7$. (See Section 15.25.)

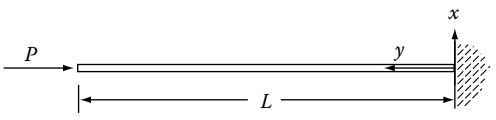


FIGURE 13.14 Column compression.

REFERENCES

1. Ogden, R. W., *Non-linear Elastic Deformations*, 219, Dover Publications, New York, 1997.
2. Simo, J. C., and T. J. R. Hughes, *Computational Inelasticity*, Springer-Verlag, New York, 1998.
3. Mooney, M., A theory of large elastic deformation, *J. Appl. Phys*., 6, 582–592, 1940.
4. Rivlin, R. S., Forty years of nonlinear continuum mechanics, *Proceedings of the IX International Congress on Rheology*, Mexico, pp. 1–29, 1984.
5. Blatz, P. J., and W. L. Ko, Application of finite elastic theory to the deformation of rubbery materials, *Trans. Soc. Rheol.*, 6, 223–251, 1962.
6. Kao, B. G., and L. Razgunas, On the determination of strain energy functions of rubbers, *Proceedings of the VI International Conference on Vehicle Structural Mechanics*, Detroit, MI, pp. 145–154, 1986.
7. Yeoh, O. H., Characterization of elastic properties of carbon-black-filled rubber vulcanizates, *Rubber Chem. Technol.*, 63, 792, 1990.
8. Wempner, G. A., Discrete approximations related to nonlinear theories of solids, *Int. J. Solids Struct.*, 7, 1581–1599, 1971.
9. Riks, E., The application of Newton's method to the problem of elastic stability, *J. Appl. Mech.*, 39, 1581–1599, 1972.
10. E. Riks, An incremental approach to the solution of snapping and buckling problems, *Int. J. Solids Struct.*, 15, 529–551, 1979.
11. Forde, B. W. R., and S. F. Stiemer, Improved arc length orthogonality methods for nonlinear finite element analysis, *Comput. Struct.*, 27, 625–630, 1987.

BIBLIOGRAPHY

Bathe, K. J., *Finite Element Procedures*, Prentice-Hall, Englewood Cliffs, NJ, 1996.

Dill, E. H., *Continuum Mechanics: Elasticity, Plasticity, Viscoelasticity*, CRC Press, Boca Raton, FL, 2007.

Holzapfel, G. A., *Nonlinear Solid Mechanics*, Wiley, New York, 2000.

Ogden, R. W., *Non-Linear Elastic Deformations*, Dover Pub., Mineola, NY, 1997.

14 Constraints and Contact

14.1 APPLICATION OF CONSTRAINTS

We often encounter situations in which the degrees of freedom **D** are not independent or they are limited in range. This is often expressible as a constraint of the form of n linear equations:

$$\mathbf{CD} - \mathbf{Q} = \mathbf{0}, \tag{14.1}$$

where matrices **C** and **Q** are constant. For example, consider the axially loaded rod in which one element is modeled as a rigid body (Figure 14.1). This condition can be applied as a constraint on the nodal displacements:

$$u_1 - u_2 = 0. \tag{14.2}$$

The constraints could be applied directly to the assembled finite element equations to reduce the number of independent variables. However, it is more convenient to incorporate the constraints 14.1 indirectly in order to provide for the general situation. There are two commonly used methods to do this: Lagrange multipliers and the penalty method.

14.1.1 Lagrange Multipliers

In the Lagrange multiplier method, the potential energy of the unconstrained structure with a nonrigid element is modified to include the constraints. For constraints of the form 14.1, set

$$\mathbf{d} = \mathbf{CD} - \mathbf{Q}. \tag{14.3}$$

The potential energy of the unconstrained structure is

$$\Pi_0 = \frac{1}{2}\mathbf{D}^{\mathrm{T}}\mathbf{KD} - \mathbf{D}^{\mathrm{T}}\mathbf{F}. \tag{14.4}$$

This includes the members that are to be constrained. That is, member 1–2 in Figure 14.1 is to be included as an elastic element.

The potential energy is augmented by

$$\Pi_c = \boldsymbol{\lambda}^{\mathrm{T}}\mathbf{d}. \tag{14.5}$$

The matrix $\boldsymbol{\lambda}$ is a diagonal matrix of parameters λ_i, which are additional independent variables called Lagrange multipliers.

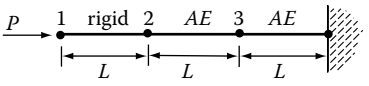


FIGURE 14.1 Constrained rod.

The modified potential energy is then

$$\begin{aligned}\Pi &= \Pi_0 + \Pi_c \\ &= \frac{1}{2}\mathbf{D}^T\mathbf{K}\mathbf{D} - \mathbf{D}^T\mathbf{F} + \boldsymbol{\lambda}^T\mathbf{d}.\end{aligned} \tag{14.6}$$

The condition that the potential energy be stationary becomes

$$\begin{aligned}\delta\Pi &= [\delta\mathbf{D}]^T\mathbf{K}\mathbf{D} - [\delta\mathbf{D}]^T\mathbf{F} + [\delta\boldsymbol{\lambda}]^T\mathbf{d} + \boldsymbol{\lambda}^T\delta\mathbf{d} \\ &= [\delta\mathbf{D}]^T\mathbf{K}\mathbf{D} - [\delta\mathbf{D}]^T\mathbf{F} + [\delta\boldsymbol{\lambda}]^T[\mathbf{C}\mathbf{D} - \mathbf{Q}] + \boldsymbol{\lambda}^T\left[\mathbf{C}\delta\mathbf{D}\right] \\ &= [\delta\mathbf{D}]^T\mathbf{K}\mathbf{D} - [\delta\mathbf{D}]^T\mathbf{F} + [\delta\boldsymbol{\lambda}]^T[\mathbf{C}\mathbf{D} - \mathbf{Q}] + [\delta\mathbf{D}]^T\left[\mathbf{C}^T\boldsymbol{\lambda}\right] = 0,\end{aligned} \tag{14.7}$$

leading to the set of equations

$$\begin{aligned}\mathbf{K}\mathbf{D} + \mathbf{C}^T\boldsymbol{\lambda} &= \mathbf{F}, \\ \mathbf{C}\mathbf{D} - \mathbf{Q} &= \mathbf{0}.\end{aligned} \tag{14.8}$$

They can be combined into the single modified finite element equation:

$$\begin{bmatrix}\mathbf{K} & \mathbf{C}^T \\ \mathbf{C} & \mathbf{0}\end{bmatrix}\begin{bmatrix}\mathbf{D} \\ \boldsymbol{\lambda}\end{bmatrix} = \begin{bmatrix}\mathbf{F} \\ \mathbf{Q}\end{bmatrix}. \tag{14.9}$$

The coefficient matrix is still symmetric but it is singular because of the zeros on the diagonal. However, if Gaussian elimination is used to solve the equations, the diagonal fills in and a solution can be constructed in the usual manner. Formally, since the inverse of the stiffness matrix for the supported structure is invertible, with **A** defined by

$$\mathbf{A} = \mathbf{C}\mathbf{K}^{-1}\mathbf{C}^T, \tag{14.10}$$

we have

$$\begin{aligned}\boldsymbol{\lambda} &= \mathbf{A}^{-1}(\mathbf{C}\mathbf{K}^{-1}\mathbf{F} - \mathbf{Q}), \\ \mathbf{D} &= \mathbf{K}^{-1}(\mathbf{F} - \mathbf{C}^T\boldsymbol{\lambda}).\end{aligned} \tag{14.11}$$

In the example shown in Figure 14.1, ignoring the constraints, and assigning the same AE to element 1–2,

$$\mathbf{K} = \begin{bmatrix} k & -k & 0 \\ -k & 2k & -k \\ 0 & -k & 2k \end{bmatrix}, \quad \mathbf{D} = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} P \\ 0 \\ 0 \end{bmatrix}. \tag{14.12}$$

The rigid constraint of member 1–2 is expressed by

$$\mathbf{C} = \begin{bmatrix} 1 & -1 & 0 \end{bmatrix}, \ \mathbf{Q} = \mathbf{0}. \tag{14.13}$$

The modified finite element equations 14.8 are therefore

$$\begin{bmatrix} k & -k & 0 & 1 \\ -k & 2k & -k & -1 \\ 0 & -k & 2k & 0 \\ 1 & -1 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ \lambda \end{bmatrix} = \begin{bmatrix} P \\ 0 \\ 0 \\ 0 \end{bmatrix}. \tag{14.14}$$

The solution by Gaussian elimination is

$$\begin{aligned} \lambda &= P, \\ u_3 &= \frac{P}{k}, \\ u_2 &= \frac{P}{k} + u_3 = 2\frac{P}{k}, \\ u_1 &= \frac{P}{k} - \frac{\lambda}{k} + u_2 = u_2. \end{aligned} \tag{14.15}$$

Note that the Lagrange multiplier can be interpreted as the reactive force due to the imposed constraint. This is also evident from the potential energy expression 14.6, where the term $\boldsymbol{\lambda}^{\mathrm{T}}\mathbf{d}$ can be regarded as the potential for the "forces" $\boldsymbol{\lambda}$, or from the virtual work expression 14.7 by the term $\boldsymbol{\lambda}^{\mathrm{T}}\delta\mathbf{d}$.

14.1.2 Perturbed Lagrangian Method

The zero diagonal element in the Lagrange multiplier method can be removed by the *perturbed Lagrangian* method. In this method, the potential (14.5) for the constraint is altered as follows.

$$\Pi_c = \boldsymbol{\lambda}^T\mathbf{d} + \frac{1}{2\kappa}\boldsymbol{\lambda}^T\boldsymbol{\lambda}. \tag{14.16}$$

Then,

$$\delta\Pi = = [\delta\mathbf{D}]^T[\mathbf{KD} - \mathbf{F}] + [\delta\boldsymbol{\lambda}]^T[\mathbf{CD} - \mathbf{Q}] + [\delta\mathbf{D}]^T\left[\mathbf{C}^T\boldsymbol{\lambda}\right] - \frac{1}{\kappa}[\delta\boldsymbol{\lambda}]^T\boldsymbol{\lambda}. \tag{14.17}$$

For the example shown in Figure 14.1, when the last term is merged into 14.14 we have

$$\begin{bmatrix} k & -k & 0 & 1 \\ -k & 2k & -k & -1 \\ 0 & -k & 2k & 0 \\ 1 & -1 & 0 & \frac{1}{\kappa} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ \lambda \end{bmatrix} = \begin{bmatrix} P \\ 0 \\ 0 \\ 0 \end{bmatrix}. \tag{14.18}$$

The last equation can now be used to eliminate the Lagrange multiplier,

$$\lambda = \kappa(u_2 - u_1), \tag{14.19}$$

and the remaining equations are

$$\begin{bmatrix} k-\kappa & -k+\kappa & 0 \\ -k+\kappa & 2k-\kappa & -k \\ 0 & -k & 2k \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} P \\ 0 \\ 0 \end{bmatrix}. \tag{14.20}$$

The solution is

$$u_3 = \frac{P}{k}, \quad u_2 = 2u_3, \quad u_1 = u_2 \frac{1 - \frac{3k}{2\kappa}}{1 - \frac{k}{\kappa}}. \tag{14.21}$$

For infinite κ, the exact result is obtained. For $\kappa = k \times (10)^3$, we obtain an approximate solution:

$$u_1 = 0.9994995\, u_2 \tag{14.22}$$

The zero on the diagonal has been removed, but the solution is now only approximate.

14.1.3 Penalty Functions

In this method, the addition to the potential energy is

$$\Pi_p = \frac{1}{2}\mathbf{d}^T\boldsymbol{\alpha}\mathbf{d}, \tag{14.23}$$

where α is a diagonal matrix of constants α_i called penalty numbers. The modified potential energy is

$$\Pi = \frac{1}{2}\mathbf{D}^T\mathbf{K}\mathbf{D} - \mathbf{D}^T\mathbf{F} + \frac{1}{2}\mathbf{d}^T\boldsymbol{\alpha}\mathbf{d}. \tag{14.24}$$

The condition that the potential energy is stationary becomes

$$\begin{aligned}\delta\Pi &= [\delta\mathbf{D}]^T\mathbf{K}\mathbf{D} - [\delta\mathbf{D}]^T\mathbf{F} + [\delta\mathbf{d}]^T\boldsymbol{\alpha}\mathbf{d} \\ &= [\delta\mathbf{D}]^T\mathbf{K}\mathbf{D} - [\delta\mathbf{D}]^T\mathbf{F} + [\mathbf{C}\delta\mathbf{D}]^T\boldsymbol{\alpha}[\mathbf{C}\mathbf{D} - \mathbf{Q}] \\ &= [\delta\mathbf{D}]^T[\mathbf{K}\mathbf{D} + \mathbf{C}^T\boldsymbol{\alpha}\mathbf{C}\mathbf{D} - \mathbf{F} - \mathbf{C}^T\boldsymbol{\alpha}\mathbf{Q}] = \mathbf{0}.\end{aligned} \tag{14.25}$$

Therefore,

$$[\mathbf{K} + \mathbf{C}^T\boldsymbol{\alpha}\mathbf{C}]\mathbf{D} = [\mathbf{F} + \mathbf{C}^T\boldsymbol{\alpha}\mathbf{Q}]. \tag{14.26}$$

The solution of this equation for very large $\boldsymbol{\alpha}$ provides an approximate solution to the constrained finite element equations.

For the example presented in Figure 14.1, we have

$$\mathbf{C}^T\boldsymbol{\alpha}\mathbf{C} = \alpha\begin{bmatrix}1\\-1\\0\end{bmatrix}\begin{bmatrix}1 & -1 & 0\end{bmatrix} = \alpha\begin{bmatrix}1 & -1 & 0\\-1 & 1 & 0\\0 & 0 & 0\end{bmatrix}. \tag{14.27}$$

The penalized equations 14.26 become

$$\begin{bmatrix}k+\alpha & -k-\alpha & 0\\-k-\alpha & 2k+\alpha & -k\\0 & -k & 2k\end{bmatrix}\begin{bmatrix}u_1\\u_2\\u_3\end{bmatrix} = \begin{bmatrix}P\\0\\0\end{bmatrix}. \tag{14.28}$$

The solution is

$$u_3 = \frac{P}{k},$$
$$u_2 = \frac{P}{k} + u_3 = 2\frac{P}{k}, \tag{14.29}$$
$$u_1 = \frac{P}{k+\alpha} + u_2.$$

If $\alpha \to \infty$, $u_1 \to u_2$. Naturally, we have to choose finite values of α for machine calculations. For $\alpha = k(10)^3$, $u_1 - u_2 = 0.00999\dfrac{P}{k}$.

Note that the contribution of member 1–2 to the stiffness matrix of the unconstrained structure is

$$\mathbf{k}^{(1-2)} = \begin{bmatrix} k & -k & 0 \\ -k & k & 0 \\ 0 & 0 & 0 \end{bmatrix}. \tag{14.30}$$

The contribution of the penalty terms to the global stiffness matrix is

$$\mathbf{k}^{(p)} = \begin{bmatrix} \alpha & -\alpha & 0 \\ -\alpha & \alpha & 0 \\ 0 & 0 & 0 \end{bmatrix}. \tag{14.31}$$

We see that the penalty method is equivalent to inserting a spring of stiffness α between the constrained nodes.

Comparing the two methods, we can see that the number of equations to be solved is increased by the number of Lagrange multipliers, but the solution is exact (to the degree of accuracy of the equation solver); however, the solver must be able to handle zero diagonal elements. In the penalty method, the number of equations to be solved is not changed, but the stiffness matrix is modified and the solution is approximate. The process of solving the equations involves the small difference of large numbers so that the equations may become poorly conditioned for very large penalty numbers and this limits the achievable accuracy.

14.1.4 Augmented Lagrangian Method

It is sometimes convenient to combine the penalty method with the Lagrange multipliers to obtain the *augmented Lagrangian* method. The modified potential energy is

$$\Pi = \frac{1}{2}\mathbf{D}^{\mathrm{T}}\mathbf{K}\mathbf{D} - \mathbf{D}^{\mathrm{T}}\mathbf{F} + \boldsymbol{\lambda}^{\mathrm{T}}\mathbf{d} + \frac{1}{2}\mathbf{d}^{\mathrm{T}}\boldsymbol{\alpha}\mathbf{d}. \tag{14.32}$$

Combining 14.14 and 14.28, we obtain the following equations for the example (Figure 14.1):

$$\begin{bmatrix} k+\alpha & -k-\alpha & 0 & 1 \\ -k-\alpha & 2k+\alpha & -k & -1 \\ 0 & -k & 2k & 0 \\ 1 & -1 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ \lambda \end{bmatrix} = \begin{bmatrix} P \\ 0 \\ 0 \\ 0 \end{bmatrix}. \tag{14.33}$$

The solution is

$$\lambda = P, \quad u_1 = u_2 = 2u_3 = 2\frac{P}{k}. \tag{14.34}$$

In this example, there is no advantage. The penalty number α drops out and the exact solution is obtained.

14.2 CONTACT PROBLEMS

Problems involving the contact between two bodies, such as an automobile tire upon the pavement or the bearing of a bolt against the bolt hole, are far more complicated than the single body problems so far considered. The actual contact surface is usually not known in advance, and the boundary conditions on this unknown surface involve tractions and displacements that are not known in advance. Consequently, the contact problem is governed by a system of inequalities and one must proceed in a step-by-step or an iterative manner. The formulation will again make use of the Lagrange multiplier method and the penalty method. In this chapter, we will only introduce the basic concepts of the analysis. ANSYS provides the option of a pure penalty method, a pure Lagrangian method, the augmented Lagrangian method, or the Lagrangian multiplier on the contact normal and the penalty factor on the frictional direction.

14.2.1 Example: A Truss Contacts a Rigid Foundation

This class of problems is illustrated by the deformation of the truss (Figure 14.2) that comes into contact with a rigid foundation when the load is large enough.

The constraint condition is

$$g - V \geq 0. \tag{14.35}$$

We will analyze the case when the member stiffness AE/L is 25 for member 1 and 50 for member 2. The finite element equation for the supported structure is

$$\mathbf{K}\begin{bmatrix} U \\ V \end{bmatrix} = \begin{bmatrix} F_x \\ F_y \end{bmatrix} \tag{14.36}$$

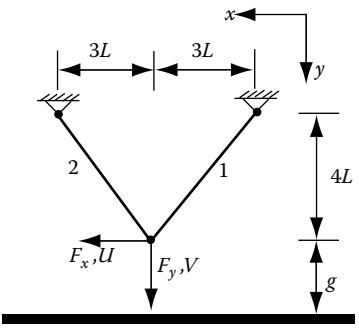


FIGURE 14.2 Truss and foundation.

with

$$\mathbf{K} = \begin{bmatrix} 27 & -12 \\ -12 & 48 \end{bmatrix}. \tag{14.37}$$

Before contact with the rigid foundation,

$$\begin{aligned} U &= \frac{F_x}{24} + \frac{F_y}{96}, \\ V &= \frac{F_x}{96} + \frac{3F_y}{128}. \end{aligned} \tag{14.38}$$

Figure 14.3 shows the nodal forces before contact. After contact at $U = U_c$ and $V = g$, with increasing loads, the node may stick or it may slide on the foundation. The foundation provides a force p resisting penetration and a force f resisting sliding (Figure 14.4). The equilibrium equations are

$$\begin{aligned} F_x - f &= 27U - 12g, \\ F_y - p &= -12U + 48g. \end{aligned} \tag{14.39}$$

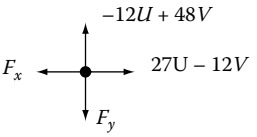


FIGURE 14.3 Equilibrium before contact.

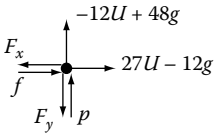


FIGURE 14.4 Equilibrium after contact.

For the stick condition, $U = U_c$ so that p and f are determined by the equilibrium equations for given loads that are large enough to maintain contact. The load and displacement at contact depends on the load path. We will consider two examples.

14.2.1.1 Load $F_y > 0$ Is Applied with $F_x = 0$

Contact ($V = g$) occurs for

$$F_y = \frac{128g}{3} \equiv F_c \tag{14.40}$$

at

$$U = \frac{4}{9}g \equiv U_c. \tag{14.41}$$

If $F_x = 0$ after contact, any additional load in the y direction is transmitted directly to the foundation without deformation of the structure. Figure 14.5 shows the nodal forces in this case: $F_y > F_c$ and $F_x = 0$. The foundation exerts a reaction force $p = F_y - F_c$ and $f = 0$. If $F_y = F_c$ after contact, a load in the x direction cannot increase the V displacement. The structure may stick to the target (no additional U displacement); or, if F_x is large enough, it may slip along the support ($U > 0$). The foundation may resist the slip (U displacement) by friction between the structure and the rigid foundation, and then a formula for f is required. We will only consider the so-called Coulomb friction: $f = \mu p$. Figure 14.6 shows the nodal forces for $F_y = F_c$ and $F_x > 0$ for slip with no friction. The nodal forces for slip including a friction force $f = \mu p$ are shown in Figure 14.7. In this case, $F_x = (27 + 12\mu)(U - U_c)$.

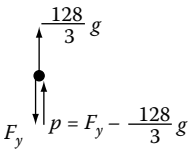


FIGURE 14.5 After contact $F_y > F_c$.

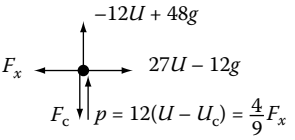


FIGURE 14.6 Slip without friction.

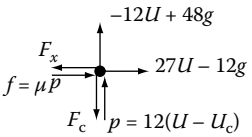


FIGURE 14.7 Slip with friction force.

14.2.1.2 Loads Are Ramped Up Together: $F_x = 27\alpha$, $F_y = 12.8\alpha$

Contact occurs for

$$\begin{aligned} \alpha &= 1.7204g, \\ F_x &= 46.4516g, \\ F_y &= 22.0215g, \\ U &= 2.1649g. \end{aligned} \tag{14.42}$$

After contact, the node may stick without further deformation and additional loads are reacted by the foundation. Or the node may slip along the foundation. If the loads are increased in the same ratio with no resistance to slip ($f = 0$), the equilibrium equations 14.39 give

$$\begin{aligned} U &= \frac{F_x + 12g}{27}, \\ p &= F_y + 12U - 48g. \end{aligned} \tag{14.43}$$

If there is a resistance to slip for which $f = \mu p$, the equilibrium equations 14.39 yield

$$\begin{aligned} U &= \frac{F_x - \mu F_y + (12 + 48\mu)g}{27 + 12\mu}, \\ p &= F_y + 12U - 48g. \end{aligned} \tag{14.44}$$

We will now formulate the problem using Lagrange multipliers and penalty functions to impose the constraint.

14.2.2 Lagrange Multiplier, No Friction Force

After contact, the constraint on the normal displacement is

$$g_{\mathrm{n}} = g - V = 0 \tag{14.45}$$

and the addition to the potential energy of the unconstrained structure is

$$\Pi_c = \lambda_{\mathrm{n}} g_{\mathrm{n}} = \lambda(g - V). \tag{14.46}$$

The first variation provides the additions to the virtual work:

$$\delta\Pi_c = \delta\lambda(g - V) + \lambda(-\delta V). \tag{14.47}$$

The modified finite element relation 14.36 is therefore

$$\begin{bmatrix} 27 & -12 & 0 \\ -12 & 48 & -1 \\ 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} U \\ V \\ \lambda \end{bmatrix} = \begin{bmatrix} F_x \\ F_y \\ -g \end{bmatrix}. \tag{14.48}$$

The equations are to be solved subject to the condition

$$\lambda(g - V) = 0. \tag{14.49}$$

That is, for $g - V < 0$ we have $\lambda = 0$, and for $g - V = 0$ we have to find λ by solving the matrix equation. Before contact, given $F_x = 0$ and $F_y > 0$, the solution is

$$\lambda = 0, \quad U = \frac{F_y}{96}, \quad V = \frac{3F_y}{128}, \tag{14.50}$$

as in 14.38. At initial contact,

$$\lambda = 0, \quad V = g, \quad U = \frac{4}{9}g, \quad F_y = \frac{128}{3}g = F_{\mathrm{c}}, \tag{14.51}$$

as in 14.40 and 14.41. If the loads are ramped up, $F_x = 27\alpha$, $F_y = 12.8\alpha$, after contact $V = g$ and we have

$$\begin{aligned} U &= \frac{F_x + 12g}{27}, \\ \lambda &= F_y + 12U - 48g. \end{aligned} \tag{14.52}$$

This agrees exactly with 14.43 and shows $\lambda = p$. Initial contact occurs at $\lambda = 0$, which leads to the result 14.42. The solution for this case is set forth in Section 15.32.

14.2.2.1 Stick Condition

Given $F_x = 0$ and $F_y > F_c$, the solution is

$$\begin{aligned} U &= \frac{4}{9}g, \\ V &= g, \\ \lambda &= -\left(F_y - \frac{128}{3}g\right) < 0. \end{aligned} \tag{14.53}$$

Comparing this result with Figure 14.5, we identify the Lagrange multiplier as the reaction force of the foundation on the structure: $\lambda = -p$.

14.2.2.2 Slip Condition

Given $F_x > 0$ and $F_y = \frac{128}{3}g$, the solution is

$$\begin{aligned} V &= g, \\ U &= \frac{F_x}{27} + \frac{4}{9}g, \\ \lambda &= -12\left(U - \frac{4}{9}g\right) = -\frac{12}{27}F_x, \end{aligned} \tag{14.54}$$

as shown in Figure 14.6. The Lagrange multiplier provides the reaction force preventing penetration of the support during slip: $\lambda = -p$.

14.2.3 Lagrange Multiplier, with Friction

The amount of slip is

$$g_t = U - U_c, \quad U_c = \frac{4}{9}g. \tag{14.55}$$

The addition to the potential energy is

$$\Pi_c = \lambda_n g_n + \lambda_t g_t \tag{14.56}$$

and the addition to the virtual work formula is

$$\begin{aligned}\delta\Pi_c &= \delta\lambda_n g_n + \lambda_n \delta g_n + \delta\lambda_t g_t + \lambda_t \delta g_t \\ &= \delta\lambda_n (g - V) + \lambda_n (-\delta V) + \delta\lambda_t (U - U_c) + \lambda_t \delta U.\end{aligned} \tag{14.57}$$

The modified finite element equation is

$$\begin{bmatrix} 27 & -12 & 0 & +1 \\ -12 & 48 & -1 & 0 \\ 0 & -1 & 0 & 0 \\ +1 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} U \\ V \\ \lambda_n \\ \lambda_t \end{bmatrix} = \begin{bmatrix} F_x \\ F_y \\ -g \\ +U_c \end{bmatrix}. \tag{14.58}$$

14.2.3.1 Stick Condition

For $U = U_c$, $V = g$, $F_y = \frac{128}{3} g$, $F_x > 0$, the solution is

$$\begin{aligned}\lambda_n &= 0, \\ \lambda_t &= F_x.\end{aligned} \tag{14.59}$$

The Lagrange multiplier λ_t is therefore the reaction of the support to prevent slip.

14.2.3.2 Slip Condition

The normal reaction on the structure from the support is $p = -\lambda_n$. The Lagrange multiplier λ_t is the friction force exerted by the foundation on the structure, $f = \lambda_t$. Assuming Coulomb friction, $f = \mu p = -\mu\lambda_n$. The virtual work of the two reaction forces is

$$\begin{aligned}\delta\Pi_c &= \delta\lambda_n (g - V) + \lambda_n (-\delta V) + f\delta U \\ &= \delta\lambda_n (g - V) + \lambda_n (-\delta V) + (-\mu\lambda_n)\delta U,\end{aligned} \tag{14.60}$$

in place of 14.57. The finite element equations become

$$\begin{bmatrix} 27 & -12 & -\mu \\ -12 & 48 & -1 \\ 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} U \\ V \\ \lambda_n \end{bmatrix} = \begin{bmatrix} F_x \\ F_y \\ -g \end{bmatrix}. \tag{14.61}$$

Note that the modified stiffness matrix is not symmetric. For $V = g$, $F_y = \frac{128}{3}g$, $F_x > 0$, $U > U_c$, the solution is

$$\begin{aligned} \lambda_n &= -12(U - U_c), \\ F_x &= 27(U - U_c) + 12\mu(U - U_c). \end{aligned} \tag{14.62}$$

That is, the applied force F_x is balanced by the internal nodal force $27(U - U_c)$ plus the resistance due to friction $\mu(12)(U - U_c)$ as shown in Figure 14.7.

14.2.4 Penalty Method

In the penalty method, the potential energy is modified by adding the potential energy of "springs" with extensions g_n and g_t:

$$\Pi_p = \frac{1}{2}\varepsilon(g_n)^2 + \frac{1}{2}\varepsilon(g_t)^2 \tag{14.63}$$

with first variation

$$\begin{aligned} \delta\Pi_p &= \varepsilon g_n \delta g_n + \varepsilon g_t \delta g_t \\ &= \varepsilon(g - V)(-\delta V) + \varepsilon(U - U_c)\delta U. \end{aligned} \tag{14.64}$$

The penalty factor ε can be different for each factor, but here it is chosen to be the same for both. Merging the terms from 14.64 into the finite element equation 14.36, the finite element equation becomes

$$\begin{bmatrix} 27+\varepsilon & -12 \\ -12 & 48+\varepsilon \end{bmatrix} \begin{bmatrix} U \\ V \end{bmatrix} = \begin{bmatrix} F_x + \varepsilon U_c \\ F_y + \varepsilon g \end{bmatrix}. \tag{14.65}$$

The solution is

$$\begin{bmatrix} U \\ V \end{bmatrix} = \frac{1}{|\mathbf{K}|} \begin{bmatrix} 48+\varepsilon & 12 \\ 12 & 27+\varepsilon \end{bmatrix} \begin{bmatrix} F_x + \varepsilon U_c \\ F_y + \varepsilon g \end{bmatrix}, \tag{14.66}$$

$$|\mathbf{K}| = \varepsilon^2 + 75\varepsilon + 1152 = 1{,}076{,}152.$$

Note that the constraint is not applied directly. The displacements are calculated for each load case, and the constraint will appear in the solution (approximately).

14.2.4.1 Stick Condition

For the load case $F_x = 0$ and $F_y > F_c$, choosing $\varepsilon = 1000$, the solution is

$$\begin{aligned} V &= g + 0.0009543(F_y - F_c), \\ U &= U_c + 0.00001115(F_y - F_c). \end{aligned} \tag{14.67}$$

Naturally, because of round-off error the first terms will not be returned exactly by the computer. We see that constraint $V = g$ and contact point $U = U_c$ are only approximated. There is some penetration of the target, as shown by the second term. The value of ε is arbitrary but must be large enough so that the penetration is negligible.

14.2.4.2 Slip Condition

If there is a resistance to slip due to Coulomb friction, the virtual work expression 14.64 must be altered to express the work of the friction force $f = \mu p = -\mu\varepsilon g_n$:

$$\begin{aligned} \delta\Pi_p &= \varepsilon g_n \delta g_n + f\delta g_t \\ &= \varepsilon(g - V)(-\delta V) - \mu\varepsilon(g - V)\delta U. \end{aligned} \tag{14.68}$$

Merging with the equation for the unconstrained structure, we have

$$\begin{bmatrix} 27 & -12 + \mu\varepsilon \\ -12 & 48 + \varepsilon \end{bmatrix} \begin{bmatrix} U \\ V \end{bmatrix} = \begin{bmatrix} F_x + \mu\varepsilon g \\ F_y + \varepsilon g \end{bmatrix}. \tag{14.69}$$

Note that the modified stiffness is not symmetric because of the friction force. The solution for $\varepsilon = (10)^4$ and $\mu = 0.5$ is

$$\begin{aligned} V &= g + 0.001194(U - U_c) + 0.0000995(F - F_c), \\ F_x &- 0.9928\mu(F - F_c) = 27(U - U_c) + 11.914\mu(U - U_c). \end{aligned} \tag{14.70}$$

The exact solution from 14.38 is

$$\begin{aligned} V &= g, \\ F_x &- \mu(F - F_c) = 27(U - U_c) + 12\mu(U - U_c). \end{aligned} \tag{14.71}$$

14.3 FINITE ELEMENT ANALYSIS

The contact problem is complicated because the topology of the system changes upon contact. The equilibrium equations before contact, for example, 14.38, are the standard finite element equations. The equilibrium equations after contact, for example,

14.39, are completely different and contain new unknowns. One must specify "stick" or "slip" conditions and a constitutive relation for the slip reaction *f*.

The solution procedure is necessarily incremental. After each load step, one must determine whether the bodies are in contact. A procedure must be devised to calculate the gap from information available in the finite element analysis. In the general case of two- or three-dimensional bodies, multiple nodes can come into contact with the finite elements forming the boundary of the target. Furthermore, after contact a node may continue in contact or it may move away, thereby creating a new gap. Because each load step is finite, initial contact occurs as a penetration of one body into another. Iteration is the necessary factor within the load step to reduce the penetration to an allowable magnitude. ANSYS treats the problem of calculating the contact of surfaces by overlaying on the given bodies, a pair of contact surfaces that interact with each other to prevent penetration and to allow calculation of the required pressures. One surface is regarded as the target and the other as the contacting surface. The contact pair must cover the potential contact region.

14.3.1 Example: Contact of a Cylinder with a Rigid Plane

Let us consider a long elastic cylinder compressed between two rigid clamps (Figure 14.8). The body is analyzed as a plane strain, which will apply to a long cylinder except near the ends.

The solution to this problem is known.[1] The half-contact width b is given by

$$b^2 \doteq \frac{2FR(1-\nu)}{\pi\mu}. \tag{14.72}$$

The contact pressure is

$$p = \frac{2F}{\pi b}\sqrt{1-(x/b)^2}. \tag{14.73}$$

The displacement of a rigid clamp is

$$d = \frac{F(1-\nu)}{\pi\mu}\left(\ln\left(\frac{4R}{b}\right)-\frac{1}{2}\right). \tag{14.74}$$

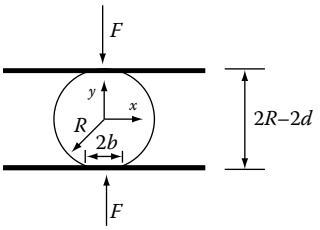


FIGURE 14.8 Compressed cylinder.

TABLE 14.1
Cylinder Contacting a Plane

For $d = 0.5$	Theory	FEA
F	25.51	25.00
B	0.1562	$0.13 < b < 0.17$
$p(0)$	104	105

This problem can be analyzed using ANSYS. The steps are given in Section 15.33 for $R = 0.5$ in., $E = 500$ lb/in. (rubber), $\nu = 0.499$. A given displacement $d = 0.05$ in. is applied and the reactive force F is calculated. The results are compared to the analytical formulas 14.72 to 14.74 in Table 14.1.

14.3.2 Hertz Contact Problem

As an example, let us consider the contact between two long cylinders pressed together by opposing forces (Figure 14.9). We suppose that the displacements are small so that linear elasticity applies. As they are pressed together, the cylinders flatten over the deformed area (Figure 14.10). The deformed position has a contact surface of width $2b$. An analytical solution to the plane strain equations is known for $b \ll R$ and assuming that the contact surface is approximately plane:[2]

$$b^2\left(\frac{1}{R_1}+\frac{1}{R_2}\right)=4F\left(k_1+k_2\right),$$
$$k=\frac{1-\nu^2}{\pi E}. \tag{14.75}$$

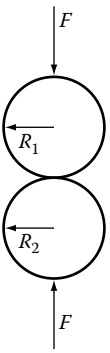


FIGURE 14.9 Contact of two cylinders.

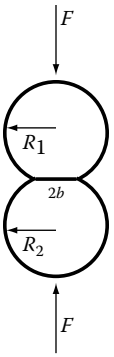


FIGURE 14.10 Contact surface.

F is the force per unit length. The contact pressure between the two cylinders is

$$p = \frac{2F}{\pi b^2}\sqrt{b^2 - x^2}, \tag{14.76}$$

where x is measured from the center of the contact line. Note that this is inherently a nonlinear problem even for small deformations. Both b and p are nonlinear functions of F.

For contact with a plane, set $R_2 = \infty$. If the plane is rigid (i.e., a very much higher modulus), set $E_2 = \infty$ to obtain

$$b^2 = \frac{4RF(1-\nu^2)}{\pi E}. \tag{14.77}$$

We will construct a finite element solution under the following assumptions. The deformed cylinder is shown in Figure 14.11. If the applied force is distributed in exactly the same manner, the diameter AB remains plane. According to the principle of St. Venant, any other distribution of load will produce only local effects near the

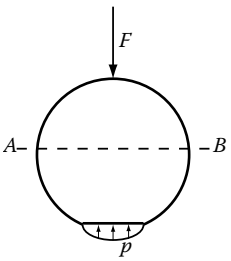


FIGURE 14.11 Contact pressure.

load. We will therefore assume that AB remains straight. We will then have double symmetry and can restrict the analysis to one-quarter of each sphere as shown in Figure 14.12. It is also easier to specify a displacement of line AB rather than force F. The magnitude of the force is then a reaction. If the displacement is not specified, there is no initial constraint on translation in the y direction. Consequently, the initial stiffness matrix for each cylinder has zero determinant because of the possible rigid body motion.

If the force is to be specified instead of the displacement, the analysis can proceed in two steps: (1) apply a very small displacement to create contact and (2) apply load F to the assembly.

ANSYS treats the problem of calculating the contact of surfaces by overlaying on the given bodies a pair of contact surfaces that interact with each other to prevent penetration and allow calculation of the required pressures. One surface is regarded as the target and the other as the contacting surface. The contact pair must cover the potential contact region (Figure 14.13).

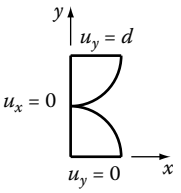


FIGURE14.12 Reduction by symmetry.

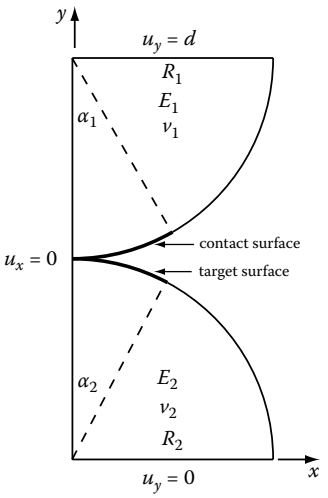


FIGURE 14.13 Contact pair model.

A possible calculation is given in Section 15.34 for $R_1 = 10$ mm, $R_2 = 13$ mm, $E_1 = E_2 = 30{,}000$ N/mm^2, $\nu_1 = \nu_2 = 0.25$. The calculated values are $F = 628$ N/mm, $p = 731$ N/mm^2, $0.419 < b < 0.558$. Given F, the expected values for p and b from 14.75 and 14.76 are $b = 0.532$ and $p = 752$, respectively.

14.4 DYNAMIC IMPACT

Typical crash problems involve dynamic impact and rebound of two objects. These problems are difficult and very computationally intense. We will demonstrate the computation by a simple example: An elastic rod moving a given velocity impacts a rigid wall (Figure 14.14). The rod is made of steel: $A = 1$ in.^{2}, $E = 2.92 \times 10^7$ lb/in.^{2}, $\rho = 0.73 \times 10^{-3}$ lbf-s^2/in.^{4}. The rod (link element) approaches the wall with initial velocity $V = 200$ in./s^2 from a gap of 0.01 in.

The exact solution is shown in Figure 14.15. The shock wave created from impact travels as a compression wave through the rod. During this time, the rod remains in contact with the rigid wall. The compression wave is then reflected as a dilatational wave upon reaching the free end of the rod and travels back to the contact surface. The rod gets separated from the rigid wall once the dilatational wave reaches the contact surface. The analysis uses node to surface contact, contact element 175, target element 169, using the default augmented Lagrangian method with increased contact stiffness FKN. The calculated result is shown in Figure 14.16. The steps in the analysis are listed in Section 15.36.

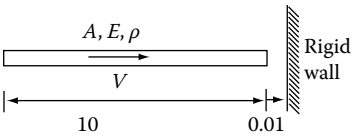


FIGURE 14.14 Rod impacting wall.

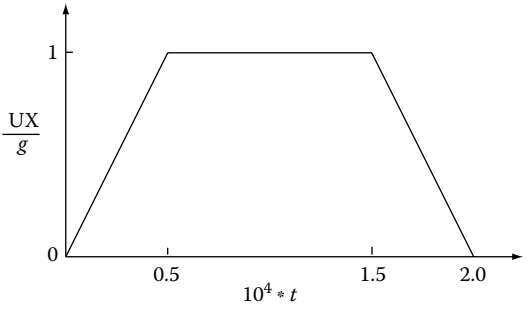


FIGURE 14.15 Exact solution for impact.

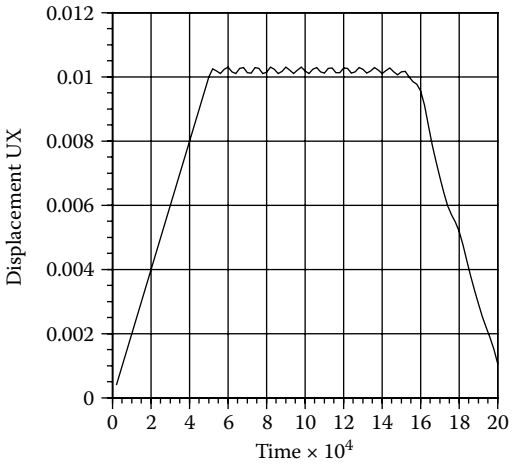


FIGURE 14.16 Calculated impact displacement.

14.5 PROBLEMS

1. Use ANSYS to determine the forces at initial contact and the final displacements for the contact of the truss (Figure 14.2) with a rigid foundation by the Lagrange multiplier method. Member 1: $A = 0.1$ lb/in., $E = 12{,}500$ lb/in. Member 2: $A = 0.2$ lb/in., $E = 12{,}500$ lb/in. $L = 10$ lb/in., $g = 0.3$ lb/in. There is no resistance to slip after contact. The loads are ramped up to $F_x = 27$ and $F_y = 12.8$. (See Section 15.33.)
2. Use ANSYS to analyze the compression of a rubber gasket modeled as a long rubber cylinder between rigid plates (Figure 14.17). The cylinder is analyzed as a plane strain problem, where $R = 0.5$ in., $E = 500$ lb/in.^{2}, $\nu = 0.499$. Since the material is nearly incompressible, use the mixed (U–P) formulation. Because of the double symmetry, only ¼ of the cylinder is needed. The plane $y = 0$ is assumed to remain plane and the nodal displacements U_Y on the plane are coupled. A vertical displacement, $d = .05$ in. of the plane is imposed. Determine the required force F, the contact region b, and the maximum contact pressure p. (See Section 15.34.)

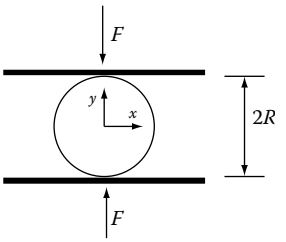


FIGURE 14.17 Compression of a rubber gasket.

3. Use ANSYS to solve the Hertz contact problem (Figure 14.9). The contact between the two long cylinders is analyzed by assuming double symmetry, Figure 14.13, for $R_1 = 10$ mm, $R_2 = 13$ mm, $E_1 = E_2 = 30{,}000$ N/mm^2, $\nu_1 = \nu_2 = 0.25$. The angle defining the potential contact region is $\alpha_1 = \alpha_2 = 8°$. Apply $d = 0.1$ and determine the total force, the maximum contact pressure, and the contact region. (See Section 15.35.)

REFERENCES

1. Gladwell, G. M. L., *Contact Problems in the Classical Theory of Elasticity*, p. 1394, Sijthoff & Noordhoff, The Netherlands, 1980.
2. Timoshenko, S., and J. N. Goodier, *Theory of Elasticity*, 2nd ed., pp. 381–382, McGraw-Hill, New York, 1951.

BIBLIOGRAPHY

Kikuchi, N., and J. T. Oden, Contact problems in elasticity, *A Study of Variational Inequalities and Finite Element Methods*, SIAM, Philadelphia, PA, 1988.

Wriggers, P., *Computational Contact Mechanics*, McGraw-Hill, New York, 1951, 2nd ed., Springer-Verlag, Berlin, 2006.

15 ANSYS APDL Examples

15.1 ANSYS INSTRUCTIONS

The instructions and examples in this chapter pertain are for version 12.1 but they are compatible with version 13. One can expect some changes with each new version. In the following instructions, C means click with the left mouse button and T means type in data, ® means ENTER. Click SAVE_DB frequently so that you can RESUM_DB if you make some unrecoverable error. Other examples of solutions using ANSYS are provided online by the HELP >ANSYS TUTORIALS button on the top toolbar.

Start the ANSYS program at your workstation. The procedure will depend on the installation on your computer system, but will be something like:

START > PROGRAMS > ANSYS 12.1 > ANSYS

A graphics window will appear as shown on the next page. A second window labeled ANSYS Output Window also appears and provides feedback throughout the analysis. It may be hidden behind the graphics window.

Actions by ANSYS are specified by specially formatted Commands as described in the Commands Reference Manual. These commands can be input from a file, typed on the command line above the graphics window, or by using the Graphics User Interface (GUI). The GUI makes it possible to select the required action by clicking on a menu item and entering the required data. Most items are adequately explained in the menus without reference to the help manuals. The general procedure in a stress analysis problem is as follows.

Preprocessor. The preprocessor module is used to create the geometric figure that is to be analyzed, and to mesh that body with prescribed element types and material properties. The elements are automatically merged to formulate the global stiffness matrix. Geometric models can also be imported from those created with CAD software such as CATIA, Parasolid, Pro/ENGINEER, SAT/ACIS, NX, SolidWorks, AutoCAD, Mechanical Desktop, and Solid Designer. The import procedure is explained in the Connection User's Guide.

Solution. The solution module is used to apply loads and support conditions, and then solve the finite element equations directly, or by iteration, or step by step in time.

Postprocessors. A postprocessor module is used to display selected results from the solution to the finite element equations such as the components of the stress tensor within an element or the components of displacement of the nodes. ANSYS provides two postprocessors. The General Postprocessor is used to examine the spatial solution at each step of the calculation. The Time History Postprocessor is used to display the time history of a variable such as the displacement of a node as a function of time. The results can be provided in text form or as a graph. Everything that

appears in the graphics window can be printed, and everything that is presented as a list can be printed.

The computer program only works with numbers. It is up to the user to input the numbers in a consistent system of units. You can use any system you like. Units are mostly not mentioned in the examples in this chapter. The examples may be regarded as dealing with a nondimensional formulation.

The ANSYS software has evolved from the work of John Swanson, who established Swanson Analysis Systems in 1970. In those days, one had to use punched cards to communicate with the computer. So the input for what became the ANSYS system was a list of instructions that contained the needed information for formulated and executing the finite element analysis—in effect, a special computer language that continues to underlay the formulation of problems by the computer mouse and the GUI. When you set up a problem today using the GUI, you are really generating a list of those special statements called commands. A list of commands can be seen in the Log file for each job, and ANSYS retains the option of input of the list of commands from a file or typing each command on the Command Line of the GUI. In order to really understand the actions within ANSYS, one must understand the ANSYS Commands.

The classic ANSYS program has been renamed to Mechanical APDL (ANSYS Parametric Design Language) application within the Workbench environment. Extensive and detailed manuals are also accessed by the HELP menu: HELP > HELP TOPICS > CONTENTS provides a list of document sets that are available. Click on MECHANICAL APDL to see a list of manuals pertaining to mechanical analysis and a brief description of each manual. The list is alphabetical and not in the order that one would normally use the manual. Of particular interest are the following manuals.

Basic Analysis Guide. This manual describes general tasks that apply to any type of analysis, including applying loads to a model, obtaining a solution, and using the ANSYS program's graphics capabilities to review results.

Theory Reference for the Mechanical APDL and Mechanical Applications. This manual provides the theoretical basis for calculations in the ANSYS program, such as elements, solvers and results formulations, material models, and analysis methods. By understanding the underlying theory, you can make better use of ANSYS capabilities while being aware of assumptions and limitations.

Element Reference. This manual describes all ANSYS elements in numerical order. It is the primary reference for the correct element type, input and output information, and comprehensive descriptions for every option for an element. It includes a pictorial catalog of the characteristics of each ANSYS element.

Command Reference. This manual describes all ANSYS commands in alphabetical order. It is the definitive reference for correct command usage, associated GUI menu paths, product applicability, and usage notes.

Structural Analysis Guide. This reference guide describes how to perform the following structural analyses: static, modal, harmonic, transient, spectrum, buckling, nonlinear, material curve fitting, gasket joint simulation, fracture, composite, fatigue, p method, beams, and shells.

Modeling and Meshing Guide. This reference manual explains how to build a finite element model and mesh it.

There are various commands to control the meshing. The major difference is between "mapped mesh" and "free mesh." A *mapped mesh* is restricted in terms of the element shape and the pattern of the mesh by selected commands. A *free mesh* has no restrictions in terms of element shapes or the pattern of mesh: these are selected by the internal meshing algorithm. However, there are commands available to place restrictions on the meshing pattern even for the *free mesh*.

15.1.1 ANSYS File Names

The Basic Guide, Chapter 18, File Management and Files, contains a list (Table 18.1) of temporary files written using the ANSYS program. Some may remain if ANSYS is terminated abnormally but have no accessible useful information. Table 18.2 lists the file extensions (jobname.xxx) of permanent files written by ANSYS. You may have to delete them manually from your directory. Following are some of the files that may occur.

The following files are ASCII text files and can be read with a word processor.

jobname.log: (log file) Every command is copied into this file. It can be used as input to repeat the sequence of commands when ANSYS is restarted.
jobname.out: (output file) This is a text file that contains all ANSYS responses to input via the GUI. It also records warnings and error messages and some results. Some of this information is also in the Output Window.
jobname.err: (error file) This is a text file containing warning and error messages.
jobname.BCS: This is a text file with performance information.
jobname.stat: This is a text file containing the status of an ANSYS batch run.
jobname.grph: (graphics file) This is an ASCII file in special format.

The following files are not text files and cannot be accessed directly. They are used by the ANSYS program.

jobname.db: (database file) This file contains all of the input data and some results. It is binary file and cannot be accessed except by the ANSYS program.
jobname.dbb: This is a binary file containing a copy of the database file created when a nonlinear analysis terminates abnormally.
jobname.esav: This is a binary file with element saved data created by a nonlinear analysis.
jobname.full: This is a binary file containing the assembled global stiffness and mass matrices.
jobname.mntr: A binary file of modal data used for a restart of a mode superposition or transient analysis.
jobname.rst: (structural analysis results) This is a binary file.
jobname.rth: (thermal analysis results) This is a binary file.
jobname.emat: (element matrices) This is a binary file.
jobname.ldhi: A binary file of load case data.
jobname.rdb: A binary file used for a multiframe restart.

15.1.2 Graphic Window Controls

15.1.2.1 Graphics Window Logo

If the ANSYS logo gets in the way, it can be removed:

PlotCtrls > Window Controls > Window Options
set LOGO = Txt in Legend

15.1.2.2 Display of Model

To change the background to white background and gray elements:

PlotCtrls > Style > Colors
select Reverse Video
PlotCtrls > Numbering >
Node Numbers ON
/NUM set to Numbers Only

To change to white elements with a black outline

PlotCtrls > Style > Colors
Picked Entity Colors
Clab set to WHITE
Lab set to Elements
OK
Pick All
Plot Ctrls > Numbering
/NUM = Numbers and Colors

15.1.2.3 Display of Deformed and Undeformed Shape White on White

Plot Ctrls >Style > Colors
check Reverse Video
PlotCtrls > Numbering >
Elem/Attrib = Element numbers
/NUM = Colors and Numbers
PlotCtrls > Style > Colors
Entity Colors
Color Assignment = User Specified
Select Elements
Select User Specified Color = WHITE
Set Entity Range = All (in first box)

15.1.2.4 Adjusting Graph Colors

PLOT CTRLS > STYLE > COLORS > GRAPH COLORS
Select colors for curve number 1, etc.

15.1.2.5 Printing from Windows Version of ANSYS

Method 1: Printing is reverse video (gray elements on a black background print as gray elements on a white background)
C PRINTER LOGO (select printer)
C OK (capture to printer)
Method 2: Printing is reverse video
PLOT CTRLS > CAPTURE IMAGE
FILE > PRINT (select printer)
C PRINT
Method 3: Printing is reverse video
PLOT CTRLS > HARD COPY > TO PRINTER (select printer)
C PRINT
Method 4: Maximum flexibility
PLOT CTRLS > HARD COPY > TO FILE

This brings up a menu where you can select a file name, file type (e.g., JPEG) and options including reverse video. The file is saved in your directory.

Print the file from your directory, not from ANSYS.

15.1.2.6 Some Useful Notes

To get white elements and black numbers on a white background for a report, you can create black elements and white numbers on a black background and then use print method 1 or method 2.

Sometimes one needs to find get node number at a particular location. There is a way to do this without scrolling through a long list of nodes. ANSYS has a function that maps coordinates to node numbers:

C UTILITY MENU > PARAMETERS > SCALAR PARAMETERS

nodenum = node(x,y,z)	[enter in selection box with actual coordinate values x,y,z]
C ACCEPT	[under items, the value of nodenum appears]

You can enter either the node number or the parameter "nodenum" on the picking menu.

Sometimes one would like to capture a snapshot of the current screen for a report. You can press the "Print Screen" button next to the right of the F12 key and then open WORD and paste the picture into a document.

15.2 ANSYS ELEMENTS SURF153, SURF154

ANSYS does not provide for direct input of tangential loads to solid elements. One has to make use of their surface effect elements: SURF153 for 2D problems and SURF154 for 3D problems. These elements have a variety of other uses including

modeling of surface tension. A description of them can be found in the Theory Reference manual. They can be viewed as a thin skin glued to the solid body that adds terms into the force stiffness and mass matrices.

SURF 153 is laid on the edge of the plane figure and is therefore a line element with the shape functions for a line element. SURF 154 is laid on the surface of a 3D element and is therefore a 2D element with the shape functions used for plane elements.

ANSYS calculates and merges the stiffness and force matrices derived from the following formulas.

$$\mathbf{K} = k^{\mathrm{f}} \int \mathbf{N}^{\mathrm{T}}\mathbf{N}\,\mathrm{d}A, \tag{15.1}$$

where k^{f} is intended to represent the stiffness of an elastic support and is input as the parameter EFS in the Real Constants menu (default is zero).

$$\mathbf{M} = \rho \int t_{\mathrm{h}}\mathbf{N}\mathbf{N}^{\mathrm{T}}\,\mathrm{d}A + A_{\mathrm{d}} \int \mathbf{N}\mathbf{N}^{\mathrm{T}}\,\mathrm{d}A, \tag{15.2}$$

where ρ is the surface element density and is input as a material property set, t_{h} is the thickness of the surface element input in the Real Constants menu (default zero), and A_{d} provides added surface mass, input as ADMSUA in the Real Constants menu (default zero).

$$\mathbf{F} = \int \mathbf{N}^{\mathrm{T}}\mathbf{p}\,\mathrm{d}A. \tag{15.3}$$

The "pressure" load $\mathbf{p}$ is directed in the local (x,y,z) directions according to the value of the parameter LKEY on the pressure load menu. LKEY = 2 denotes a load along x-axis, LKEY = 3 denotes load along y-axis, and LKEY = 1 denotes a load along z-axis (normal). Refer to the manual for other options for the load and for a damping matrix.

15.3 TRUSS EXAMPLE

In this example, the nodes and elements are created by direct input of coordinates and node numbers.

For the truss shown in Figure 15.1, given $A = 1$, $L = 1$, $E = 1$, $FX_1 = 1$, $FY_1 = 0$, we will calculate the displacements of node 1.

a. Set job name and preferences.

Login and start ANSYS APDL	[wait for graphics window]
C FILE > CHANGE JOBNAME	[top menu bar]
T a job name	[in place of "file"]
Select NEW LOG Yes	
OK	

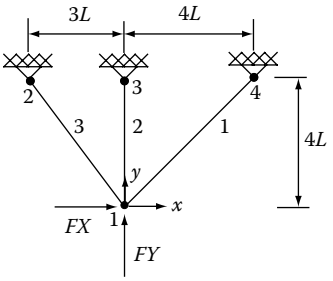


FIGURE 15.1 Truss.

C PREFERENCES
C STRUCTURAL
C OK

b. Select element and input real constants.

C PREPROCESSOR > ELEMENT TYPE > ADD > ADD
C LINK > 3D finit str 180 [this is the truss element]
C OK
C CLOSE
C REAL CONSTANTS > ADD > ADD
C OK
T 1 [for the Area]
C OK
C CLOSE

c. Input material properties.

C MATERIAL PROPERTIES > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1 [for EX, the modulus of elasticity]
T 0.3 [for PRXY, the Poisson ratio, not actually used for a truss element]
C OK
C MATERIAL > EXIT [from materials window]

d. Create nodes and elements.

C MODELING > CREATE > NODES > IN ACTIVE CS
Enter NODE = 1, x,y,z = 0,0,0 [coordinates of node 1]
C APPLY
T 2 –3 4 0 [coordinates of node 2]
C APPLY
T 3 0 4 0 [coordinates of node 3]
C APPLY
T 4 4 4 0 [coordinates of node 4]
C OK [see 4 nodes]

C LIST > NODES [on top menu bar]
C OK
C FILE > CLOSE [if nodes are correct]
C CREATE > ELEMENTS
AUTO NUMBERED > THRU NODES [picking menu appears]
C on nodes 1 and 4 [element 1]
C APPLY [in picking window]
C on nodes 1 and 3 [element 2]
C APPLY
C on nodes 1 and 2 [element 3]
C OK [see three elements]
C LIST > ELEMENTS > NODES + ATTRIBUTES [top menu bar]
C FILE > CLOSE [if elements are correct]

e. Input loads and supports.

C LOADS > DEFINE LOADS > APPLY > STRUCTURAL
C FORCE > ON NODES
C on node 1
C OK [in picking window]
Select LAB = FX [should already be selected]
Enter VALUE = 1
C OK [see load vector]
C STRUCTURAL DISPLACEMENT > ON NODES
C on nodes 2,3,4
C OK
C ALL DOF
Enter VALUE = 0
C OK [see supports]

f. Solve equations.

C SOLUTION > SOLVE > CURRENT LS
CLOSE status window
C OK in solve window
CLOSE "solution is done" window

g. Examine results.

C GENERAL POST PROCESSOR
C LIST RESULTS > NODAL SOLUTION > DOF SOLUTION > X-COMP DISPL
C OK [see UX = 6.2397; this file can be printed for the record]
C CLOSE
C LIST RESULTS > NODAL SOLUTION > DOF SOLUTION > Y-COMP DISPL
C OK [see UY = 0.10183; this file can be printed for the record]
C FILE > CLOSE
C LIST RESULTS > REACTION SOLU [ALL ITEMS selected]
C OK [see list of reactions at nodes 2,3,4; this file can be printed for the record]

C FILE > CLOSE
C QUIT [middle menu bar]
C OK

15.4 BEAM BENDING

In this example, the element data table is used to extract the element stress.

Consider a cantilever beam with an end load (Figure 15.2). The beam has a rectangular cross section: $h = 2$, $w = 1$, $A = 2$, $I = 2/3$, $E = 1000$, $L = 5$, $P = 1$. Use two elements as shown in Figure 15.2 to calculate the displacements, moments, and bending stress.

a. Activate ANSYS and set preferences. [default job name is "file"]
See example 15.3.

b. Choose elements and materials.
C PREPROCESSOR
enter ET,1,BEAM3 on command line [BEAM3 Element]
enter KEYOPT,1,6,1 on command line [of data on member forces and moments]
enter R,1,2,2/3,2 on command line for Area, IZZ, and h
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
Enter 1000 for EX parameter
C in NPRXY box
Enter 0.3 for PRXY parameter [the Poisson ratio is actually not used]
C OK
C MATERIAL > EXIT

c. Create keypoints at the ends and connect them by a line.
C PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
Select Element Numbers from the ELEM/ATTRIB numbering menu
C OK
C PREPROCESSOR > MODELING > CREATE > KEYPOINTS > IN ACTIVE CS
Enter POINT 1 at 0,0,0

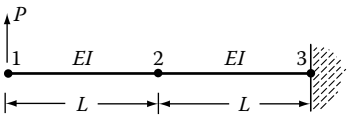


FIGURE 15.2 Bending of a beam.

C APPLY
Enter POINT 2 at 10,0,0
C OK
C PREPROCESSOR > MODELING > CREATE > LINES > LINES
C STRAIGHT LINE
C on keypoints
C OK in picking menu

d. Mesh the line.

C PREPROCESSOR > MESHING
C SIZE CONTROLS
C MANUAL SIZE
C GLOBAL > SIZE
Set NDIV = 2
C OK
C MESH
C LINES
Pick line
C OK [see line with two elements and three nodes]

e. Apply support conditions and loads.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON NODES
C node 2 [right end]
C OK
C ALL DOF
C on box for VALUES
T 0 for the value of the displacements and rotation
OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C FORCE/MOMENT
C ON NODES
C node 1 [left end]
C OK
Select FY [for LAB]
C on VALUE box
T value for FY = 1
C OK [force shown on the node]

f. Set solution parameters and sole equations.

C UNABRIDGED MENU
C SOLUTION > LOAD STEP OPTIONS > OUTPUT CONTROLS
C SOLUTION PRINT OUT [to store data on element forces]
C ALL ITEMS [from Item Menu]
C LAST SUBSTEP
C OK
C SOLUTION > SOLVE > CURRENT LS
C FILE > CLOSE status window

C OK in solve window
C CLOSE solution window [Solution is done]

g. View results.

C GENERAL POSTPROCESSOR
C LIST RESULTS
C NODAL SOLUTION
C DOF SOLUTION
Select Y-COMP of Displacement
C OK [see list of node displacements: UY = 0.5 at node 1]
C FILE > CLOSE
C LIST RESULTS
C NODAL SOLUTION
C DOF SOLUTION
Select Z-COMP of rotation
C OK [see list of rotations at each node: −0.075 at node 1]
C FILE > CLOSE
C GENERAL POSTPROCESSOR
C ELEMENT TABLE
C DEFINE TABLE
C ADD
Enter user label = M
Select ITEM = BY SEQ NO [scroll down to find this choice]
Select SMISC
Enter SMISC, 12 [this is a code number for moment at the right end of the element]
C OK [see the element reference manual for the codes for BEAM3]
C CLOSE
C GENERAL POSTPROCESSOR > LIST RESULTS
C ELEM TABLE DATA
Select M
C OK [see M = 5 for element 1 and M = 10 for element 2]
C FILE > CLOSE
C GENERAL POSTPROCESSOR > ELEMENT TABLE > DEFINE TABLE
C ADD
Enter user label = S
Select ITEM = BY SEQ NO [scroll down to find this choice]
Select LS
Enter LS,6 [this is a code number for bending stress at the right end of the element]

C OK
C CLOSE
C GENERAL POSTPROCESSOR > LIST RESULTS
C ELEM DATA TABLE [deselect M]
Select S
C OK [see S = 15 for element 2]
C FILE > CLOSE
C QUIT
C OK

15.5 BEAM WITH A DISTRIBUTED LOAD

In this example, a nonuniform pressure load is introduced.

A cantilever beam with a linearly varying load (Figure 15.3) and a rectangular cross section has the following dimensions: $h = 1$, $w = 1$, $A = 1$, $I = 1/12$, $E = 30 \times 10^6$, $L = 10$, $p = 1$. We are to calculate the displacements. One element gives the exact solution: $\text{UY} = \dfrac{11pL^4}{120EI} = \dfrac{11}{30} \times 10^{-4}$ downward at the end.

a. Activate ANSYS and set preferences.
See example 15.3.

b. Choose elements and materials.
C PREPROCESSOR
enter ET,1,BEAM3 on command line [BEAM3 Element]
enter R,1,1,1/12,1 on command line for Area, IZZ, and h
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
Enter 30e6 for EX parameter
C in NPRXY box
Enter 0.3 for PRXY parameter [the Poisson ratio is actually not used]
C OK
C MATERIAL > EXIT

c. Create nodes and elements.
C PREPROCESSOR > MODELING > CREATE > NODES > IN ACTIVE CS
Enter POINT 1 at 0,0,0

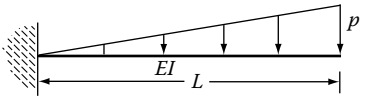


FIGURE 15.3 Bending of a beam with distributed load.

C APPLY
Enter POINT 2 at 10,0,0
C OK
C PREPROCESSOR > MODELING > CREATE > ELEMENTS
C AUTO-NUMBERED > THRU NODES
C on node 1 then node 2 [beam element: $I = 1$, $J = 2$]
C OK in picking menu

d. Apply support conditions and loads.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON NODES
C node 1 [left end, or Type 1 in box on picking menu]
C OK
C ALL DOF
C on box for VALUES
T 0 [for the value of the displacements and rotation]
OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C PRESSURE > ON BEAMS
C on beam element [or Type 1 in box on picking menu]
C OK [load menu appears]
Enter LKEY = 1, VALI = 0, VALJ = 1 [load varies linearly from node I to node J]
C OK [representative force shown on the element]

e. Solve the equations.

C SOLUTION > SOLVE > CURRENT LS
C CLOSE status window
C OK in solve window
C CLOSE solution window [solution is done]

f. Extract results.

C GENERAL POSTPROCESSOR
C LIST RESULTS
C NODAL SOLUTION
C DOF SOLUTION
Select Y-COMP of Displacement
C OK [see list of node displacements: UY = -0.36667×10^{-3} at node 2]
C FILE > CLOSE
C QUIT
C OK

15.6 ONE TRIANGLE

In this example, nodes are created by specifying coordinates and elements are created by picking nodes.

The analysis is to check the element stiffness matrix for a plane stress triangle (Figure 15.4) as generated by the Plane 182 element. In nondimensional units, $E = 16$, $\nu = 1/3$, $\mu = 6$.

a. Activate ANSYS and set preferences.
See example 15.3.

b. Select element and material properties.
C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
C 4 NODE 182 [the triangle is a special case of this element]
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 16 for EX parameter
C in PRXY box
T 1/3 for PRXY parameter [ANSYS converts this to five significant figures]
C OK [$\mu = 6$]
C MATERIAL > EXIT

c. Create nodes and elements.
C PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
C OK
C PREPROCESSOR > MODELING > CREATE > NODES > IN ACTIVE CS
T node = 1, x = 0, y = 0
C APPLY
T node = 2, x = 1, y = 0
C APPLY

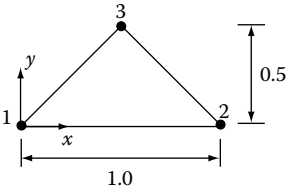


FIGURE 15.4 Plane stress triangle.

T node = 3, x = 0.5, y = 0.5
C OK
C PREPROCESSOR > MODELING > CREATE > ELEMENT >
C AUTO NUMBERED > THRU NODES
C nodes 1,2,3 [specifying only three nodes implies a triangle]
C OK

d. Apply boundary conditions.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON NODES
C nodes 2 and 3
C APPLY
C ALL DOF if it is not selected already
C on box for VALUES
T 0 for the value of the displacement components
C APPLY
C node 1
APPLY
C UY [deselect ALL DOF]
C APPLY
C node 1
OK
C UY [to deselect UY]
C UX
C on VALUE box
T value for UX of 1
C OK
C LIST > LOADS > DOF CONSTRAINTS [Utility menu at top]
C ON ALL LINES [check boundary conditions]
C FILE > CLOSE

e. Solve for displacements and stresses.

C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window
C OK in Solve window
C YES in warning window [ANSYS suggests reduced integration]
C CLOSE on information window [solution is done]

f. Display results.

C GENERAL POST PROC > LIST RESULTS > REACTION SOLU
C OK to get a list of nodal forces that comprise column one of the stiffness matrix. [this can be compared with column 1 of $\mathbf{k}^1$, Equation 4.45]
C FILE > CLOSE
C QUIT
C OK

15.7 PLANE STRESS EXAMPLE USING TRIANGLES

This example illustrates the use of the COPY command to create nodes and elements and the box method in the picking menu.

The given problem is the short beam (Figure 15.5). The solution for the corner displacement is tabulated in Table 4.3. In nondimensional units: $a = 1$, $p = 1$, $E = 1$, $\nu = 1/3$.

a. Activate ANSYS and set preferences.
See example 15.3.

b. Choose element and material.
C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
C 4 NODE 182 [PLANE182 element]
C OK [the constant strain triangle is a special case this element]
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1 for EX parameter [nondimensional parameters]
C in PRXY box
T 1/3 for PRXY parameter [PR is Poisson ratio]
C OK
C MATERIAL > EXIT

c. Create nodes and elements.
C PREPROCESSOR > MODELING > CREATE
C NODES > IN ACTIVE CS
T 1 0 0 0 [for Node,X,Y,Z]
C APPLY

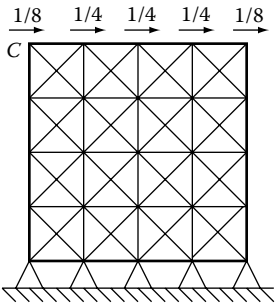


FIGURE 15.5 Short beam, 64 triangles.

T 2 0.25 0 0 [for Node,X,Y,Z]
C APPLY
T 11 0.25 0.25 0 [for Node,X,Y,Z]
C APPLY
T 10 0 0.25 0 [for Node,X,Y,Z]
C APPLY
T 6 0.125 0.125 0 [note the numbering, chosen for the copy command]
C OK
C PREPROCESSOR > MODELING > CREATE
C ELEMENTS > AUTO NUMBERED > THRU NODES
C nodes 1 then 2 then 6 [specifying three nodes implies a triangle]
APPLY
C nodes 2 then 11 then 6
APPLY
C nodes 11 then 10 then 6
APPLY
C nodes 10 then 1 then 6
OK
C PREPROCESSOR > MODELING > COPY
C ELEMENTS > AUTO NUMBERED
C PICK ALL
T 4 for ITIME [leave NINC = 1]
T 0.25 for DX [creating bottom row of elements]
C OK
C PREPROCESSOR > MODELING > COPY
C ELEMENTS > AUTO NUMBERED
C PICK ALL
T 4 for ITIME
T 9 for NINC
T 0 for DX
T 0.25 for DY
C OK [creating columns of elements]
C PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
C OK

d. Apply boundary conditions.

C PREPROCESSOR > LOADS > DEFINE LOADS > APPLY
C STRUCTURAL > DISPLACEMENT > ON NODES
C BOX [on picking menu]
DRAW A BOX AROUND THE BOTTOM NODES BY CLICK AND DRAG
C OK
C ALL DOF [if it is not selected already]
C on box for VALUES

T 0 for the value of the displacement components
C OK [supports shown on the bottom edge]
C FORCE > ON NODES
C upper right and upper left corner nodes
C APPLY
C FX if it is not already selected
C on VALUE box
T value for FX of 1/8
C APPLY
C all intermediate nodes along the top [38, 39, 40]
C OK
T value for FX of 1/4
C OK [forces shown on the nodes]

e. Solve for displacements and stresses.
C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window
C OK in Solve window
C YES TO CONTINUE SOLUTION [ANSYS rightly warns against triangles]
C CLOSE on information window [solution is done]

f. Display results.
C GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see deformed shape
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C DOF SOLUTION
C X-COMPONENT OF DISPLACEMENT
C OK to get a list of nodal displacements [UX = 6.7341 at node 41]
C FILE > CLOSE [this list can be printed]

15.8 CANTILEVER BEAM MODELED AS PLANE STRESS

This example introduces the function editor for application of nonuniform loading and the Query tool for investigating the stress distribution.

This beam of Example 15.5 is analyzed as a plane stress problem as shown in Figure 15.6.

The body has a unit depth and $L = 10$, $h = 1$, $w = 1$, $p = 1$, $E = 30 \times 10^6$, $\nu = 0.27$. It is modeled as a plane stress problem. The solution by elementary beam theory, which

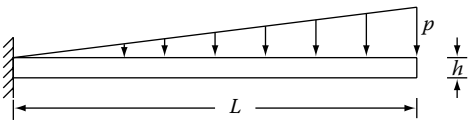


FIGURE 15.6 Cantilever beam as plane stress.

neglects shear deformations, is a tip displacement with value $11pL^4/120\,EI = 0.367 \times 10^{-3}$ and a maximum bending stress of $Mc/I = 2pL^2/wh^2 = 200$.

a. LOGIN and launch ANSYS.
See example 15.3.

b. Define elements and materials.
C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
Select 8 NODE 183 [PLANE183 element]
C OK [plane stress is the default]
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 30e6 for EX parameter
C in PRXY box
T 0.27 for PRXY parameter [PR is Poisson ratio]
C OK
C MATERIAL > EXIT

c. Create geometric figure.
C PREPROCESSOR > MODELING > CREATE
C AREAS > RECTANGLE > BY DIMENSIONS
T 0 10 0 1 for X1,X2,Y1,Y2
C OK and the region appears

d. Mesh figure.
C PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
C OK
C PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [mesh using default size]
[2 high by 15 wide mesh appears with node numbers]

e. Define load distribution function.
C PARAMETERS > FUNCTIONS > DEFINE/EDIT [on top utility menu]
C 0.1*
Select X in the drop menu currently showing TIME [Result = 0.1*{X}]
C GRAPH [click INV if GRAPH is not a current label in the gray buttons]
Enter X-Axis Range: 0 to 10
C GRAPH [see graph of function]
C CLOSE Plot Information window [may be behind graphics window]

C FILE > SAVE [in Function Editor window]
Enter File name PSR [saved as PSR.func]
C SAVE
C FILE > CLOSE
C PARAMETERS > FUNCTIONS > READ FROM FILE
Enter PSR.func
C OPEN
Enter Table parameter name = PRESS
C OK
C PLOT > ELEMENTS

f. Apply boundary conditions.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON LINES
C left edge of the model
C OK
C ALL DOF if it is not selected already
C on box for VALUES
T 0 for the value of the displacement components
C OK [supports shown on the left edge]
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C PRESSURE > ON LINES
C top line [$Y = 1$]
C OK
Select EXISTING TABLE from SFL menu
OK
Select PRESS
OK [arrow appears to indicated pressure load]

g. Solve finite element equations.

C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window
C OK in Solve window
C FILE > CLOSE on information window [solution is done]

h. Display results.

C GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see deformed shape
C PLOT > ELEMENTS
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C DOF SOLUTION
C Y-COMPONENT
C OK to get a list of nodal displacements [UX = -0.36893×10^{-3} at node 34]
C FILE > CLOSE
C GENERAL POST PROC > QUERY RESULTS > SUBGRID SOLU
C STRESS
C X-DIRECTION

C OK
C node 36 [maximum bending stress = 202.57]
C OK [not accurate with this few elements]

15.9 PLANE STRESS: PURE BENDING

In this example, the use of symmetry and antisymmetry to apply boundary conditions is introduced.

The problem diagrammed in Figure 15.7. One-quarter of the body is analyzed using symmetry conditions. In nondimensional units: $a = 2$, $b = 2$, $p = 1$, $E = 1$, $\nu = 0.3$. To prevent rigid displacement we set $u(0,0) = v(0,0) = 0$. An approximate solution is obtained by using a four-node element as described in Section 5.1.2.

a. Activate ANSYS and set the job name and preferences.
See example 15.3.

b. Choose element and material.
C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
C 4 NODE 182 [Plane182 element]
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1 for EX parameter
C in PRXY box
T 0.3 for PRXY parameter
C OK
C MATERIAL > EXIT

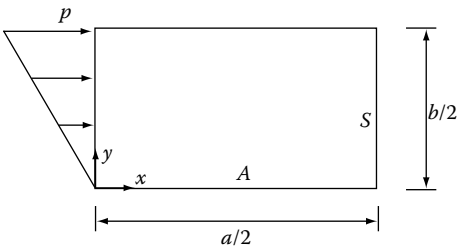


FIGURE 15.7 Plane stress: pure bending.

c. Create geometric figure.
C PREPROCESSOR > MODELING > CREATE
C AREAS > RECTANGLE > BY DIMENSIONS
T 0 1 0 1 for X1,X2,Y1,Y2
C OK and the region appears

d. Mesh figure.
C PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
C ELEMENT NUMBERS from Elem pull-down menu
C NUMBERS ONLY from /NUM pull-down menu
C OK
C PREPROCESSOR > MESHING > SIZE CONTROLS
C MANUAL SIZE > LINES > PICKED LINES
Pick top and bottom
C APPLY
C box for NDIV
T 2 in NDIV box to mesh with 2 divisions per side
C APPLY
Pick left and right ends
C OK
T 2 in NDIV box to mesh with 2 divisions per side
C OK
C PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [2 × 2 grid of square elements]

e. Define load distribution functions.
C PARAMETERS > FUNCTIONS > DEFINE/EDIT [on top utility menu]
C 1*
Select Y in the drop menu currently showing TIME [Result = 1*{Y}]
C FILE > SAVE [in Function Editor window]
Enter File name: SIGX [saved as SIGX.func]
C SAVE
C FILE > CLOSE
C PARAMETERS > FUNCTIONS > READ FROM FILE
Select SIGX.func
C OPEN
Enter Table parameter name = SIGX
C OK

f. Apply boundary conditions.
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON NODES
C on node at the origin
C OK

Select ALL DOF [UX = 0 and UY = 0 at the origin]
C on box for VALUES
T 0 for the value of the displacement components
C OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C SYMMETRY B.C. > ON LINES
C RIGHT EDGE
C OK
C ANTISYMM B.C. > ON LINES
C BOTTOM EDGE
C OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
PRESSURE > ON LINES
C on the left edge
C OK
Select EXISTING TABLE from SFE drop down menu
C OK [check that SIGX is selected]
C OK [pressure equals a compressive stress]
[you can check on the loads by LIST > LOADS > SURFACE > ON ALL LINES]

g. Get solution.
C SOLUTION > SOLVE > CURRENT LS
C FILE > CLOSE status window
C OK in Solve window
C CLOSE on information window [solution is done]

h. Display results.
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C STRESS
C X-COMPONENT
C OK [SX = −0.98985 at node 4, SX = −0.49029 at node 5]
C FILE > CLOSE
C QUIT
C OK

15.10 PLANE STRAIN BENDING EXAMPLE

The SURF element is introduced for application of shear loads. A change in material properties without repeating the entire analysis is also demonstrated.

This example is a solution of the problem described in Section 5.1.3. The complete structure is shown in Figure 15.8, but the analysis uses only the upper half and antisymmetry on the line $y = 0$. In nondimensional terms: $c = 2$, $L = 16$, $P = 32$, $E = 8192$, $\nu = 0.3$.

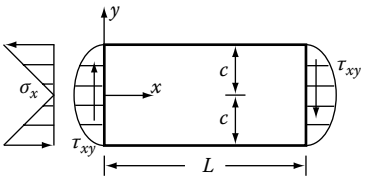


FIGURE 15.8 Plane strain bending example.

a. Activate ANSYS and set jobname and preferences.
See example 15.3.

b. Choose elements and materials.
C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
C 4 NODE 182 [Plane182 element]
C OK
C OPTIONS
C PLANE STRAIN on the K3 drop down menu
C OK
C ADD
C SURFACE EFFECT
2D SURF 153 should be selected
OK
SELECT SURF153
C OPTIONS
FOR K4 SELECT EXCLUDE MIDSIDE NODES
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
CC STRUCTURAL
CC LINEAR
CC ELASTIC
CC ISOTROPIC
T 8192 for EX parameter
C in PRXY box
T 0.3 for PRXY parameter
C OK
C MATERIAL > EXIT

c. Create geometry.
C PREPROCESSOR > MODELING > CREATE
C AREAS > RECTANGLE > BY DIMENSIONS
T 0 16 0 2 for X1,X2,Y1,Y2
C OK and the region appears

d. Mesh figure.

C PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
Select ELEMENT NUMBERS
Select NUMBERS ONLY
C OK
C PREPROCESSOR > MESHING > SIZE CONTROLS
C MANUAL SIZE > LINES > PICKED LINES
Pick top and bottom
C APPLY
C box for NDIV
T 8 in NDIV box to mesh with 8 divisions per side
C APPLY
Pick left and right ends
C OK
T 4 in NDIV box to mesh with 4 divisions per side
C OK
PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [creates 4 × 8 mesh]
MODELING > CREATE > ELEMENTS > ELEM ATTRIBUTES
Select TYPE = 2 SURF 153
C OK
SURF/CONTACT > SURF TO SURF
C OK (Tlab = Top surface)
C BOX [on picking menu]
Draw box around nodes on RIGHT edge by holding down mouse button and dragging
C APPLY
C OK
C BOX
Draw box around nodes on LEFT edge by holding down the mouse button and dragging
C OK
LIST > ELEMENTS > NODES
Check the I and J nodes of new elements of type 2 [down on the right and up on the left]
CLOSE

e. Define load distribution functions.

PARAMETERS > FUNCTIONS > DEFINE/EDIT [on top utility menu]
C 96*
Select Y in the drop menu currently showing TIME [Result = 96*{Y}]
FILE > SAVE [in Function Editor window]

Enter File name: MOMENT [saved as MOMENT.func]
SAVE
FILE > CLOSE
PARAMETERS > FUNCTIONS > READ FROM FILE
Select MOMENT.func
OPEN
Enter Table parameter name = MOMENT
OK
PARAMETERS > FUNCTIONS > DEFINE/EDIT [on top utility menu]
C − 12 + 3 * [note minus sign]
Select Y in the drop menu currently showing TIME
C *
Again, select Y in the drop menu [Result = >12+3*{Y}*{Y}]]
C FILE > SAVE [in Function Editor window]
Enter File name = SHEAR [saved as SHEAR.func]
C SAVE
C FILE > CLOSE
C PARAMETERS > FUNCTIONS > READ FROM FILE
Select SHEAR.func
C OPEN
Enter Table parameter name = shear
C OK

f. Apply boundary conditions.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON LINES
C bottom edge of the model
C OK
Select UX
C on box for VALUES
T 0 for the value of the displacement components
C OK
C DISPLACEMENT > ON NODES
Pick node in lower left corner
C APPLY
Select UY
T value = 0
C APPLY
Pick node in upper left corner
C OK
Select UX [deselect UY]
C OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL

C PRESSURE > ON ELEMENTS
C BOX (on picking menu)
Draw a box around the surface elements on the right edge
C OK
SET LKEY = 2 [tangential load on the element]
Select EXISTING TABLE from SFE drop down menu
C OK
Select SHEAR
C OK [no indication of loading]
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C PRESSURE > ON ELEMENTS
C BOX (on picking menu)
Draw a box around the surface elements on the left edge
C OK
SET LKEY = 2 [tangential load on the element]
Select EXISTING TABLE from SFE drop down menu
C OK
Select SHEAR
C OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C PRESSURE > ON ELEMENTS
C BOX (on picking menu)
Draw a box around the surface elements on the left edge
C OK
SET LKEY = 1 [pressure load on the element]
Select EXISTING TABLE from SFL drop down menu
C OK
Select MOMENT
C OK
[you can check on the loads by LIST > LOADS > SURFACE > ON ELEMENTS]

g. Get solution.
C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window
C OK in Solve window
C CLOSE on information window [solution is done]

h. Display results.
C GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C OK and check deformed shape
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C DOF SOLUTION
C Y-COMPONENT
C OK to get a list of nodal displacements [UY = 0.85774 at lower right corner]

C FILE > CLOSE

i. Change Poisson ratio and repeat analysis.

C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C MATERIAL MODEL NUMBER 1 [in the left column]
C LINEAR ISOTROPIC
Change PRXY to 0.499 [very large bulk modulus]
OK
MATERIAL > EXIT
C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window
C OK in Solve window
C CLOSE on information window [solution is done]
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C DOF SOLUTION
C Y-COMPONENT
C OK to get a list of nodal displacements [UY = 0.74183 at the lower right corner]
C CLOSE
C QUIT

15.11 PLANE STRESS EXAMPLE: SHORT BEAM

Antisymmetry is used to implement boundary conditions in this example.

This is a solution of the example presented in Section 5.1.4 and shown in Figure 15.9. Nondimensional parameters are used: $a = 1$, $p = 1$, $E = 1$, $\nu = 0.3$. One-half of the body and symmetry conditions are used. A 4 × 8 mesh of four-node square elements is used.

a. Activate ANSYS and set preferences.

See example 15.3.

b. Choose elements and materials.

C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
C 4 node 182

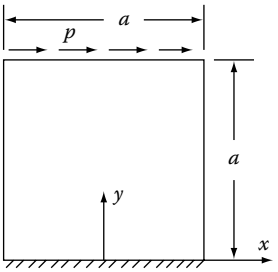


FIGURE 15.9 Plane stress example: short beam.

C OK
C ADD
C SURFACE EFFECT
2D SURF 153 should be selected
OK
SELECT SURF153
C OPTIONS
FOR K4 SELECT EXCLUDE MIDSIDE NODES
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1 for EX parameter
C in PRXY box
T 0.3 for PRXY parameter
C OK
C MATERIAL > EXIT

c. Create geometry.

C PREPROCESSOR > MODELING > CREATE
C AREAS > RECTANGLE > BY DIMENSIONS
T 0 0.5 0 1 for X1,X2,Y1,Y2
C OK and the region appears

d. Mesh figure.

C PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
C OK
C PREPROCESSOR > MESHING > SIZE CONTROLS
C MANUAL SIZE > GLOBAL > SIZE
C box for NDIV
T 0.125 in SIZE box to control element size
C OK
PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [4 × 8 square mesh appears with node numbers]
C PREPROCESSOR > MODELING > CREATE > ELEMENTS
C ELEMENT ATTRIBUTES
Select 2 SURF 153 on TYPE menu
OK
C SURF / CONTACT > SURF TO SURF
C OK [accept TLAB = TopSurface]
C BOX on picking menu
Use cursor to draw a box around the top edge
C OK in picking menu

e. Apply boundary conditions.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C DISPLACEMENT > ON LINES
C bottom edge of the model
C OK
C ALL DOF if it is not selected already
C on box for VALUES
T 0 for the value of the displacement components
C OK [supports shown on the bottom edge]
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C ANTI-SYMM > ON LINES
C left edge of the model
C OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C PRESSURE > ON ELEMENTS
C BOX in picking menu
C draw around top edge
C OK
Enter LKEY = 2
Enter VALUE = 1
C OK [close Warning window]

f. Solve for displacements and stresses.

C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window
C OK in Solve window
C CLOSE on information window [solution is done]

g. Display results.

C GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see deformed shape
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C DOF SOLUTION
C X-COMPONENT
C OK to get a list of nodal displacements [UX = 7.1207 at node 6]
C CLOSE
C GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
C STRESS
C Y-COMPONENT
C OK to get a list of stress components extrapolated to each node [SY = −7.3741 at node 2]
C CLOSE [remember that there is actually a singularity at node 2]
C QUIT
C OK

15.12 SHEET WITH A HOLE

Use of the Boolean operator to subtract areas, use of the Mesh Tool, and Refinement of the Mesh are introduced.

A sheet with a central hole (Figure 15.10) is stretched by a uniform edge stress S resulting in a stress concentration at the hole. In nondimensional variables: $S = 100$, $a = 20$, $b = 10$, $r = 5$, $E = 1$, $\nu = 0.3$.

15.12.1 Solution Procedure

a. Activate ANSYS, change job name, and set preferences to structural.
See example 15.3.

b. Choose element and material.
C PREPROCESSOR > ELEMENT TYPE > ADD
C ADD in the new window
C SOLID
C 4 node 182
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1 for EX parameter
C in PRXY box
T 0.3 for PRXY parameter
C OK
C MATERIAL > EXIT

c. Create the body by first creating a rectangle then subtracting the circular area.
C PREPROCESSOR > MODELING > CREATE
C AREAS > RECTANGLE > BY 2 CORNERS

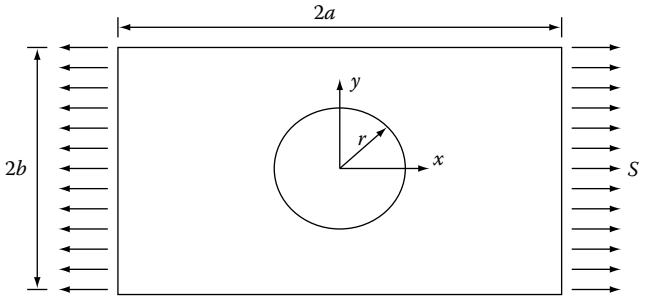


FIGURE 15.10 Sheet with a hole.

Enter WP X = 0, WP Y = 0 [this establishes the origin of the working plane]
Enter Width = 20, Height = 10 [dimensions of the rectangle]
C OK and the rectangular region appears
C PREPROCESSOR > MODELING > CREATE
C AREAS > CIRCLE > SOLID CIRCLE
Enter WP X = 0, WP Y = 0 [center of the circular area]
Enter Radius = 5
C OK [circular area appears]
C PLOT CTRLS > NUMBERING [top line menu]
C AREA NUMBERING [rectangle is A1, circle is A2]
C OK
C PREPROCESSOR > MODELING > OPERATE > BOOLEANS
C SUBTRACT > AREAS
PICK RECTANGULAR AREA
C OK
PICK CIRCLULAR AREA
C OK [body appears as area A3]

d. Mesh the body.

C PREPROCESSOR > MESHING > MESH TOOL [mesh tool menu appears]
Select Smart Size [note: slider sets at 6 by default]
C MESH [near the bottom of mesh tool menu]
C Pick All [coarse mesh appears]
C REFINE [Refine at Elements selected]
Pick elements along the arc of the circle
OK [level 1 selected]
OK [refined mesh appears]

e. Apply boundary conditions.

C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT
C SYMMETRY > ON LINES
C bottom edge and the left edge of the model
C OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
C PRESSURE > ON LINES
C RIGHT EDGE
C OK
C on VALUE box
T –100 for the value [negative pressure is tension]
C OK

f. Solve for displacements and stresses.

C SOLUTION > SOLVE > CURRENT LS
C CLOSE information window

C OK in Solve window
SELECT Yes in warning window, if any.
C CLOSE on information window [solution is done]

g. Display results.
C GENERAL POST PROC > QUERY RESULTS > SUBGRID
SOLUTION
Select Stress SX
OK
Pick points where you want the stress displayed [SX = 434 at top of circle]
OK [i.e., the stress concentration = 4.34]
PLOT > REPLOT (erases results)
Repeat for any other desired results.
C QUIT
C OK

15.13 PLASTICITY EXAMPLE

Inelastic materials, plotting of results on a graph, the contour plot, and the Time History Post-processor are introduced. We begin to omit the C for familiar commands.

This problem is a solution of the short beam problem (Figure 15.11) for an elastic–plastic material using a von Mises–type yield function and isotropic strain hardening with a bilinear stress–strain model. In lb-inch units: $a = 1$, $p = 10{,}000$, $E = 30 \times 10^6$, $\nu = 0.3$, $Y = 30{,}000$, $E_T = 2.727 \times 10^6$, $E_p = 3 \times 10^6$. The load is applied in 10 steps ($0 \le t \le 1$) and the elastic–plastic state is displayed. Then the load is removed ($1 \le t \le 2$) and the residual stress is evaluated. Symmetry is not used.

a. Activate ANSYS and set job name and analysis type.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD

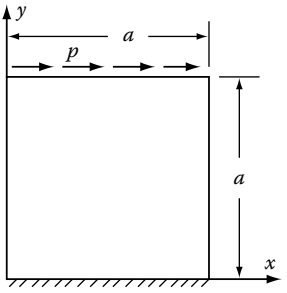


FIGURE 15.11 Short beam: plasticity.

SOLID
QUAD 4node 182
APPLY
SURFACE EFFECT > 2D STRUCTURAL 153
OK
C SURF 153
OPTIONS
FOR K4 SELECT EXCLUDE MIDSIDE NODES
OK
CLOSE
PREPROCESSOR > MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
NONLINEAR
INELASTIC
RATE INDEPENDENT
ISOTROPIC HARDENING PLASTICITY
MISES PLASTICITY
BILINEAR [request for elastic properties appears]
OK
Enter 30e6 for EX parameter
Enter 0.3 for PRXY parameter
OK
Enter 30000 for the Yield Strs
Enter 2.727e6 for the Tang Mod
OK
MATERIAL > EXIT
PLOT on the top menu
DATA TABLES
OK [plots stress–strain diagram]

c. Create geometry.

PREPROCESSOR > MODELING > CREATE
AREAS > RECTANGLE > BY DIMENSIONS
Enter 0 1 0 1 for X1,X2,Y1,Y2
OK and the region appears

d. Mesh body.

PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MESHING > MESHTOOL
SMART SIZE
Set slider to 4 [produces a 17 × 17 element mesh]
MESH [at the bottom of mesh tool menu]
PICK ALL [the coarse mesh appears]
C REFINE at Elements on mesh tool [it may be hidden behind graphics window]

Pick a fan of elements at the lower two corners 4-3-2-1 pattern
OK
OK (accept level 1) [a refined mesh of 523 quad elements appears]
CLOSE MESH TOOL
MODELING > CREATE > ELEMENTS > ELEM ATTRIBUTES
SET TYPE = 2 SURF 153
OK
MODELING > CREATE > ELEMENTS > SURF/CONTACT > SURF TO SURF
OK [Tlab = Top surface]
C BOX [on picking menu]
Draw box around the top nodes by holding down the mouse button and drag left to right
OK
PLOT CTRLS > NUMBERING [top line menu]
NODE NUMBERS OFF
SET ELEM/ATTRIB TO ELEMENT NUMBERS [to see the new elements]
SELECT NUMBERS ONLY
OK [new elements 524–540]
[Zoom in if desired by PLOT CTRLS > PAN ZOOM ROTATE or Icons on right]
LIST > ELEMENTS > NODES
Check that the new elements are type 2
FILE > CLOSE
PLOT CTRLS > NUMBERING
SET ELEMENT NUMBERING TO NO NUMBERING
OK

e. Apply boundary conditions.

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON LINES
C bottom edge of the model
OK
ALL DOF
C on box for VALUES
T 0 for the value of the displacement components
OK [supports shown on the bottom edge]
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON ELEMENTS

C BOX [on picking menu]
Draw a box around the surface elements on top edge
OK
SET LKEY = 2 [tangential load on the element]
SET VALUE = 10000 [magnitude of the distributed load along the element]
OK [close warning]

f. Solve for displacements and stresses for the first load step.

The total load is specified, then reached by ramping up in small time steps to reach the full load at TIME = 1. Results are saved after each time step.

SOLUTION > [if set to Abridged Menu]
UNABRIDGED MENU
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS
File write frequency: select "every substep"
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME AND SUBSTPS
Enter 10 as number of substeps [load ramped is default]
OK
SOLUTION > SOLVE > CURRENT LS [note time at end of step = 1]
CLOSE on information window
OK in solve window [convergence information is displayed as solution is marched out]
CLOSE on information window when solution is done

g. Remove the load and solve for the residual stress after elastic unloading.

UTILITY MENU > PLOT > ELEMENTS
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON ELEMENTS
C BOX [on picking menu]
Draw a box around the surface elements
OK
SET LKEY = 2
SET VALUE = 0
OK
SOLUTION > SOLVE > CURRENT LS [note time at end of step = 2]
CLOSE information window
OK in solve window
CLOSE information window when solution is done

h. Display results: list displacements of selected nodes after each time step.

GENERAL POST PROC > READ RESULTS > BY TIME/FREQ
Time = 1 [end of loading cycle]

OK
GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
OK and see deformed shape
UTILITY MENU > PLOT > ELEMENTS
GENERAL POST PROC > QUERY RESULTS > SUBGRID SOLUTION
STRESS SY
OK [picking menu appears]
Pick a node to see the value of SY [SY = 69,601 at lower-left corner]
OK
PLOT RESULTS > CONTOUR PLOT > NODAL SOLUTION
STRESS > von Mises stress
OK [note yielded region of S > 30,000]
UTILITY MENU > PLOT > ELEMENTS
TIME HIST POSTPROC [close information window FILE > CLOSE]
DEFINE VARIABLES [note that variable 1 is TIME, which is a magnitude of load]
ADD [click on the background to see the body]
Select Nodal DOF Result
OK
Pick upper right corner node (node 3)
OK
Enter name: U3
Select Translation UX
OK [variable 2 is UX at node 3]
Close [you may have to move windows to see the Defined Variables window]
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 2
OK [list UX at corner for each time step]
CLOSE

i. Plot stress distribution along a selected path after loading.

Utility Menu > PLOT > ELEMENTS
GENERAL POSTPROC > PATH OPERATIONS >
DEFINE PATH > ON WORKING PLANE
OK
Pick nodes at lower left then right corners [zoom in as needed]

OK
Enter path name: Base
enter nDiv = 50 [number of data points]
OK
FILE > CLOSE

GENERAL POSTPROC > PATH OPERATIONS > MAP ONTO PATH
Enter label: SYP [this is Sy after plastic loading]
Select Stress
Select Y-direction SY
OK
GENERAL POSTPROC > PATH OPERATIONS > PLOT PATH ITEM > ON GRAPH
Select SYP
OK [graph appears, graph parameters can be adjusted with PLOT CTRLS menu]
PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter y-axis label: Stress Sy
Select y-axis range: specified
Enter −70000, +70000
Enter Number of y-axis divisions: 10
OK
PLOT > REPLOT

j. Plot stress distribution along the base after unloading.

UTILITY MENU > PLOT > ELEMENTS
GENERAL POST PROC > READ RESULTS > LAST SET
GENERAL POST PROC > QUERY RESULTS > SUBGRID SOLUTION
STRESS SY
OK [picking menu appears]
Pick a node to see the value of SY [SY = 34,584 at node 2]
OK
GENERAL POSTPROC > PATH OPERATIONS > MAP ONTO PATH
Enter label: SYR [this is the residual SY after unloading]
Select Stress
Select Y-direction SY
OK
GENERAL POSTPROC > PATH OPERATIONS > PLOT PATH ITEM > ON GRAPH
Select SYR
OK [this is the residual stress after unloading]
GENERAL POSTPROC > PATH OPERATIONS > ADD
Enter LabR = SYE [SY for elastic unload]
Enter FACT1 = 1
Select Lab1 = SYP
Enter FACT2 = −1
Select Lab2 = SYR
OK [SYE = SYP − SYR]

GENERAL POSTPROC > PATH OPERATIONS > PLOT PATH ITEM > ON GRAPH
Select SYP, SYR, and SYE
OK

15.14 VISCOELASTICITY CREEP TEST

Viscoelastic materials are introduced. A step load is approximated by a very short time step. The Restart command is used to extend the solution.

This is a solution of the creep test for the standard linear solid model in shear with elastic volume change (Equation 9.17). The model for shear is as shown in Figure 15.12. In nondimensional units: $\mu_1 = \mu_2 = \eta = 100$. $G(0) = 100$, $G(\infty) = 0.5$, $G_1 = 50$, $\tau_1 = 0.5$. The volume change is elastic with bulk modulus $\kappa = 650/3$. Thus, $E_0 = 260$, $\nu_0 = 0.3$. One element in plane stress provides the exact spatial distribution since the strain and stress are uniform.

a. Activate ANSYS and set analysis type.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
Structural Solid
C 8 node 183 [this is element Plane183]
OK [plane stress is the default]
CLOSE
PREPROCESSOR > MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
LINEAR > ELASTIC > ISOTROPIC
EX = 260 [E(0)]

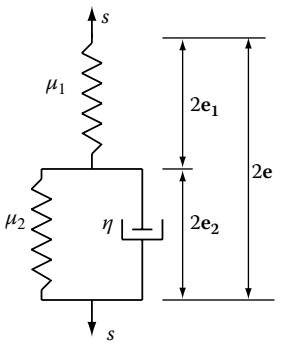


FIGURE 15.12 Viscoelastic model for shear.

PRXY = 0.3 [NU(0)]
OK
NONLINEAR
VISCOELASTIC
PRONY
SHEAR RESPONSE
a1 = 0.5 [ANSYS alpha 1]
t1 = 0.5 [ANSYS tau 1]
OK [volumetric response is elastic]
MATERIAL > EXIT

c. Create geometry.
PREPROCESSOR > MODELING > CREATE
AREAS > RECTANGLE > BY DIMENSIONS
T 0 1 0 1 for X1,X2,Y1,Y2
C OK and the region appears

d. Mesh figure.
PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MESHING > SIZE CONTROLS
MANUAL SIZE > GLOBAL > SIZE
C box for NDIV
T 1 in NDIV box to mesh with one 8-node element
[one element provides the exact solution for this problem]
OK
PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [element appears with corner node numbers]
LIST > NODES [if you want to see a list of all 8 nodes]
OK
CLOSE

e. Apply boundary conditions.
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON LINES
C bottom edge of the model
OK
Select UY
Enter 0 for the value of the displacement component
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL >
DISPLACEMENT > ON NODES
C bottom left corner of the model
OK
Select UX
Enter 0 for the value of the displacement component
OK [close warning window]

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON LINES
C upper edge of the model
OK
Enter Load PRES value −1 [negative pressure is tension]
OK

f. Solve for displacements and stresses.

The total load is specified then applied in one step. Results are saved after each time step. First, we apply the load in very small time step to approximate a step load for this creep problem.
SOLUTION > UNABRIDGED MENU
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME-TIME STEP
Time at end of load step: enter .0001
Time step size: enter .0001
Select STEPPED
OK
SOLUTION > SOLV CURRENT LS
CLOSE information window
C OK in solve window
CLOSE information window when solution is done
GENERAL POSTPROC > READ RESULTS > LAST SET
LIST RESULTS > NODAL SOLUTION
Select DOF solution
Select Y-COMPONENT
OK [Check to see if UY = 0.38463×10^{-2} at top nodes]
CLOSE

g. Next, we march out the solution maintaining the current loads constant.

SOLUTION > ANALYSIS TYPE > RESTART
Load Step Number: 1
Sub Step Number: 1
OK
CLOSE information window
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE
Item to be controlled: select “nodal DOF solu”
File write frequency: select “every substep”
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME-TIME STEP
Time at end of load step: enter 5.0
Time step size: enter 0.1
Select STEPPED

DELTIM minimum = 0.01
DELTIM maximum = 0.1
OK
SOLUTION > SOLV CURRENT LS
OK [close warning]
[convergence information is displayed as solution is marched out]
CLOSE information window when solution is done

h. Display results: list displacements of selected nodes after each time step.

GENERAL POST PROC > READ RESULTS > LAST SET
GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see deformed shape
TIME HIST POSTPROC [close information window]
DEFINE VARIABLES [new window appears showing variable 1 is the time]
Add
Select Nodal DOF Result
OK
Pick corner node 6 [move window to see model]
OK
Enter name: DISP
Select Translation UY [variable 2 is UY]
OK
Close information window
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 2
OK [list UY at corner for each time step, UY = 0.0060546 at $t = 5$]
FILE > PRINT [if desired]
CLOSE

i. Plot results: plot the displacement at each time.

Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: TIME
Enter y-axis label: DISP
SELECT: SPECIFIED Y-RANGE
ENTER Y-RANGE OF 0 TO 0.01
OK
TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable: enter 1 [time on x-axis]
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 2 [displacement on y-axis]
OK

15.15 VISCOELASTICITY EXAMPLE

This is a solution of the plane strain problem solved in Section 9.1.2 that is diagrammed in Figure 15.13. Material properties are as in Section 15.14, and the constant load is $\sigma = 1$. In nondimensional units: $\mu_1 = \mu_2 = \eta = 100$. $G(0) = 100$, $G(\infty) = 0.5$, $G_1 = 50$, $\tau_1 = 0.5$. The volume change is elastic with bulk modulus $\kappa = 650/3$. Thus, $E_0 = 260$, $\nu_0 = 0.3$. Support conditions are added to prevent rigid displacement without constricting the model.

a. Activate ANSYS and set analysis type.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
Structural Solid
C 8 node 183 [this is element Plane183]
OK
OPTIONS
Select K3 = plane strain
OK
CLOSE
PREPROCESSOR > MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
LINEAR > ELASTIC > ISOTROPIC
EX = 260 [E(0)]
PRXY = 0.3 [NU(0)]
OK
NONLINEAR
VISCOELASTIC

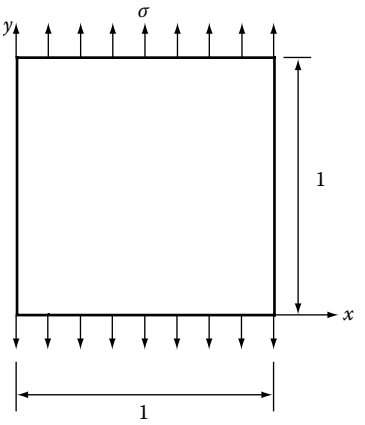


FIGURE 15.13 Viscoelastic: plane strain.

PRONY
SHEAR RESPONSE
a1 = 0.5 [ANSYS alpha 1]
t1 = 0.5 [ANSYS tau 1]
OK [volumetric response is elastic]
MATERIAL > EXIT

c. Create geometry.
PREPROCESSOR > MODELING > CREATE
AREAS > RECTANGLE > BY DIMENSIONS
T 0 1 0 1 for X1,X2,Y1,Y2
C OK and the region appears

d. Mesh body.
PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MESHING > SIZE CONTROLS
MANUAL SIZE > GLOBAL > SIZE
C box for NDIV
T 1 in NDIV box to mesh with one 8-node element
[one element provides the exact solution for this problem]
OK
PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL (element appears with corner node numbers)
LIST > NODES [if you want to see a list of all eight nodes]
OK
CLOSE

e. Apply boundary conditions.
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON LINES
C bottom edge of the model
OK
Select UY
Enter 0 for the value of the displacement component
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL >
DISPLACEMENT > ON NODES
C bottom left corner of the model
OK
Select UX
Enter 0 for the value of the displacement component
OK [close warning window]
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE >
ON LINES
C upper edge of the model
OK

Enter Load PRES value −1 [negative pressure is tension]
OK

f. First, we apply the load in very small time step to approximate a step load for this creep problem.

SOLUTION > UNABRIDGED MENU
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME-TIME STEP
Time at end of load step: enter .0001
Time step size: enter .0001
Select STEPPED
OK
SOLUTION > SOLV CURRENT LS
CLOSE information window
C OK in solve window
CLOSE information window when solution is done
GENERAL POSTPROC > READ RESULTS > LAST SET
LIST RESULTS > NODAL SOLUTION
Select DOF solution
Select Y-COMPONENT
OK [Check to see if UY = 0.35×10^{-2} at top nodes]
CLOSE

g. Next, we march out the solution maintaining the current loads constant.

SOLUTION > ANALYSIS TYPE > RESTART
Load Step Number: 1
Sub Step Number: 1
OK
CLOSE information window
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE
Item to be controlled: select “nodal DOF solu”
File write frequency: select “every substep”
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME-TIME STEP
Time at end of load step: enter 5.0
Time step size: enter 0.1
Select STEPPED
DELTIM minimum = .01
DELTIM maximum = 0.1
OK
SOLUTION > SOLV CURRENT LS
OK [close warning; convergence information is displayed as solution is marched out]
CLOSE information window when solution is done

h. Display results: list displacements of selected nodes after each time step.

GENERAL POST PROC > READ RESULTS > LAST SET
GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see deformed shape
TIME HIST POSTPROC [close information window]
DEFINE VARIABLES [new window appears showing variable 1 is the time]
Add
Select Nodal DOF Result
OK
Pick corner node 6 [move window to see model]
OK
Enter name: DISP
Select Translation UY [variable 2 is UY]
OK
Close information window
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 2
OK [list UY at corner for each time step, UY = 0.0060546 at $t = 5$]
FILE > PRINT [if desired]
CLOSE

i. Plot results: plot the displacement at each time.

Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: TIME
Enter y-axis label: DISP
SELECT: SPECIFIED Y-RANGE
ENTER Y > RANGE OF 0 TO 0.01
OK
TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable: enter 1 [time on x-axis]
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 2 [displacement on y-axis]
OK

15.16 MODE SHAPES AND FREQUENCIES OF A ROD

In this example, the determination of natural frequencies and mode shapes for the finite element model is introduced.

This is the solution to the problem analyzed in Section 10.4.2. The rod is modeled by two elements using lumped mass (Figure 15.14). The natural frequencies and

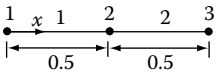


FIGURE 15.14 Axial deformation of a rod.

mode shapes for axial displacement are determined. In nondimensional units: $A = 1$, $E = 1$, $\rho = 1$, $L = 1$.

a. Activate ANSYS with structural preference.
See example 15.3.

b. Choose elements and material.
Use LINK1 element, enter the area in the Real Constants menu. Linear, elastic, isotropic material model with E = 1, NUXY is not used. Density = 1.
PREPROCESSOR > ELEMENT TYPE > ADD > ADD
C 3D finit str 180
OK
CLOSE
REAL CONSTANTS > ADD > ADD
OK
Enter AREA = 1
OK
CLOSE
MATERIAL PROPERTIES > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
Enter EX = 1
Enter PRXY = 0.3 [Poisson ratio, not used]
OK
DENSITY
Enter 1
OK
MATERIAL > EXIT

c. Create nodes and elements.
MODELING > CREATE > NODES > IN ACTIVE CS
Enter NODE = 1, x,y,z = 0,0,0
APPLY
Enter 2 0.5 0 0
APPLY
Enter 3 1 0 0
OK

ELEMENTS > AUTO NUMBERED > THRU NODES
C on nodes 1 then 2
APPLY
C on nodes 2 then 3
OK

d. Apply loads and support conditions.
SOLUTION > DEFINE LOADS > APPLY
STRUCTURAL > DISPLACEMENT > ON NODES
SELECT NODE 1 AND NODE 2
APPLY
SET UY = 0
APPLY
SELECT NODE 3
OK
SET ALL DOF = 0
OK

e. Specify analysis type and options and solve.
SOLUTION > ANALYSIS TYPE > NEW ANALYSIS
C MODAL
OK
SOLUTION > ANALYSIS TYPE > ANALYSIS OPTIONS
Enter 2 for No. of nodes to be extracted
Enter 2 for No. of modes to expand
Set LUMPM = YES [to use lumped mass]
OK (Lanczos options window appears)
OK to accept defaults [note: modes normalized to the mass matrix]
SOLUTION > SOLVE > CURRENT LS
CLOSE (in Stat window)
OK (To begin solution)
CLOSE (solution is done)

f. List natural frequencies and mode shapes.
GENERAL POSTPROC > RESULTS SUMMARY
(list of natural frequencies in CPS appears)
CLOSE window
GENERAL POSTPROC > READ RESULTS > FIRST SET
LIST RESULTS > NODAL SOLUTION > DOF SOLUTION
SELECT X COMPONENT
OK [see displacements for mode 1]
CLOSE
GENERAL POSTPROC > READ RESULTS > NEXT SET
LIST RESULTS > NODAL SOLUTION > DOF SOLUTION
SELECT X COMPONENT
OK [see displacements for mode 2]
CLOSE

15.17 MODE SHAPES AND FREQUENCIES OF A SHORT BEAM

Animation of mode shapes is introduced in this example.

This is the plane stress problem shown in Figure 15.15 and discussed in Section 10.5. In nondimensional units, $a = 1$, $b = 1$, $E = 1$, $\nu = 1/3$, $\rho = 1$. A 10×10 grid of 100 elements is used, together with a consistent mass model, to calculate the mode shapes and natural frequencies in cycles per second.

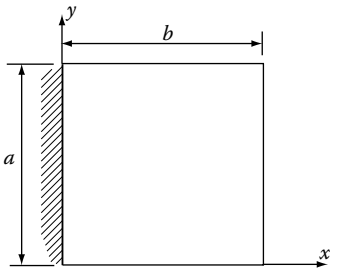


FIGURE 15.15 Plane stress: vibrations.

a. Activate ANSYS structural.
See example 15.3.

b. Choose element and material, create figure and mesh the body, apply support conditions.
Use Plane 182 element, mesh the body with NDIV = 10, and specify material properties including the density and apply support conditions.

c. Specify analysis type and options and solve.
SOLUTION > ANALYSIS TYPE > NEW ANALYSIS
C MODAL
OK
SOLUTION > ANALYSIS TYPE > ANALYSIS OPTIONS
Enter 10 for No. of nodes to be extracted
Enter 10 for No. of modes to expand
OK (Lanczos options window appears)
OK to accept defaults
SOLUTION > SOLVE > CURRENT LS
CLOSE (in Stat window)
OK (To begin solution)
CLOSE (solution is done)

d. List natural frequencies.
GENERAL POSTPROC > RESULTS SUMMARY
(list of natural frequencies in CPS appears)
CLOSE window

e. View the mode shapes.
GENERAL POSTPROC > READ RESULTS > FIRST SET
PLOTCTRLS > ANIMATE > MODE SHAPE

OK [see animated first mode shape = first bending mode]
C Stop [in animation controller window, which may be hidden behind the graphics window]
Close
GENERAL POSTPROC > READ RESULTS > NEXT SET
PLOTCTRLS > ANIMATE > MODE SHAPE
OK [see animated second mode shape = extension mode]
Stop [animation controller]
Close
[you can continue to view other modes: third is second bending mode, fourth is a breathing mode, etc.]

15.18 TRANSIENT ANALYSIS OF SHORT BEAM

Transient dynamic motion is analyzed by Newmark's method in this example.

This is the solution of the example problem discussed in Section 10.8 for transient dynamical motion due to a step load. The body is shown in Figure 15.16. In nondimensional units, $a = 1$, $b = 1$, $E = 1$, $\nu = 1/3$, $\rho = 1$, $p = 1$. Note that if the SURF element is used to apply the shear load, the density of the materials of the SURF element must be specified as zero.

a. Activate ANSYS with structural preference.
See example 15.3.

b. Choose element and material, create figure and mesh the body, apply load and support conditions.
Use the Plane182 element, mesh the body with NDIV = 10, specify material properties including the density, and apply support and load conditions as for the static problem (Section 15.11).

c. Specify analysis type and options and solve.
SOLUTION > UNABRIDGED MENU
SOLUTION > ANALYSIS TYPE > NEW ANALYSIS
C transient

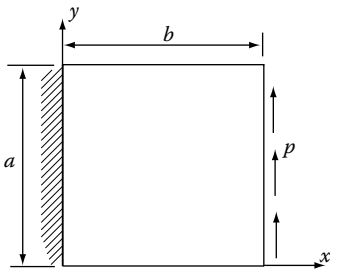


FIGURE 15.16 Short beam: transient motion.

OK [the options box appears]
OK [i.e., use Full]
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS >
DB/RESULTS FILE [unabridged menu]
Item to be controlled: select "nodal DOF solu"
File write frequency: select "every substep"
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME/TIME STEP
Time at end of load step: enter 10
Time step size: enter 0.1
Select STEPPED [i.e., load is ramped up in first step]
OK
C SOLUTION > SOLV CURRENT LS
C CLOSE on information window
C OK in solve window
C CLOSE on information window when solution is done

d. List results.

TIME HIST POSTPROC
Close information window
DEFINE VARIABLES
ADD
Select Nodal DOF Result
OK
Pick corner node
OK
Enter name: DISPLACEMENT
Select Translation Component UY
OK
Close
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 2
OK
CLOSE

e. Plot results.

Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: TIME
Enter y-axis label: DISPLACEMENT
Select: specified x-range
Enter x-range of 0 to 10
Select: specified y-range
Enter y-range of 0 to 15
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 2
OK

15.19 STRESS INTENSITY FACTOR BY CRACK OPENING DISPLACEMENT

The special crack tip element is used with automatic calculation of the stress intensity factor.

A cracked sheet is loaded in tension (Figure 11.1). Because of double symmetry, we can use one-quarter for the analysis with symmetry conditions on the edges $x = 0$ and $y = 0$ as shown in Figure 15.17. The stress intensity factor will be determined by the crack opening method. In nondimensional units: $E = 1$, $\nu = 0.3$, $S = 1$.

a. Activate ANSYS and set job name and preferences.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
C ADD
C SOLID
C QUAD 8node 183
C OK
C OPTIONS
Plane stress should be selected
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROP > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1.0 for EX parameter
C in PRXY box
T 0.3 for PRXY parameter
C OK
C MATERIAL > EXIT

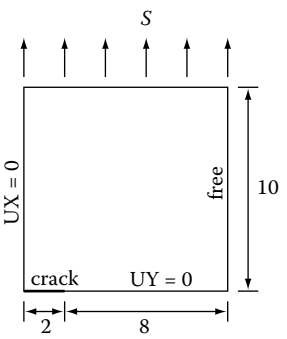


FIGURE 15.17 Cracked sheet.

c. Establish geometry and mesh the object.

The upper right quarter of the cracked sheet is used. The origin as a KEYPOINT is placed at the crack tip by using two rectangles and then combining them into one material body. The origin must be at the crack tip.

C PREPROCESSOR > MODELING > CREATE > AREAS >
C RECTANGLE > BY DIMENSIONS
T −2 0 0 10 for X1,X2,Y1,Y2 [note the minus sign]
C APPLY and the first rectangle appears
T 0 8 0 10 for X1,X2,Y1,Y2
C OK and the second rectangle appears [this ensures a keypoint at the origin]
PLOT CTRLS > NUMBERING
C box after Keypoint Numbers to turn them ON
C box after Line Numbers to turn them ON
C box after Area Numbers to turn them ON
C box after Node numbers to turn them ON
C OK (see all features numbered)
C PREPROCESSOR > MODELING > OPERATE > BOOLEANS > ADD > AREAS
C PICK ALL [this combines the two areas into one area A3]
C PREPROCESSOR > MESHING > SIZE CONTROLS
C CONCENTRATE KPs > CREATE
PICK crack tip at the origin
C OK (menu appears)
Enter .25 for radius of first row of elements
Enter 1 for ratio of second row to first
Enter 6 for no. of elements around circum.
Select SKEWED 1/4 PT from menu for KCTIP
C OK
C PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [close warning message]

d. Apply boundary conditions and solve for displacements and stresses.

PLOT > LINES
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT >
ON LINES
C bottom right edge of the model (L9)
C APPLY
C UY
Enter 0 for the value of the displacement components
C APPLY
C left edge of the model (L4)
C OK
C UX

Enter 0 for the value of the displacement components (Symmetry Condition)
C OK
C SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON LINES
C top edge of *both* of the original two elements (L3 and L10)
C OK
Enter –1 for Load Pressure value [negative for tension]
C OK
C SOLUTION > SOLV CURRENT LS
C CLOSE on information window
C OK in SOLVE window
C YES in warning window
CLOSE on information that solution is complete

e. Display results and calculate stress intensity factor.
The active origin must be at the crack tip and a path of three nodes for calculation of K must be defined.
C GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C OK and see deformed shape
C UTILITY MENU > PLOT CTRLS > PAN,ZOOM,ROTATE
Use circle and arrows to zoom in on crack tip region
C GENERAL POSTPROC > PATH OPERATIONS > DEFINE PATH > BY NODES
PICK crack tip and next two nodes on the crack surface (1, 61, 59)
C OK
Enter CRACK for path name.
C OK
C CLOSE list
C GENERAL POST PROC > NODAL CALCS > STRESS INT FACTR
Select PLANE STRESS from the menu for KPLAN
Select HALF-SYMM B.C. from the menu for KCSYM
C OK produces listing showing KI = 2.6478
CLOSE

15.20 STRESS INTENSITY FACTOR BY *J*-INTEGRAL

The Select command and use of the Command Line for typed commands are introduced.

A cracked sheet is loaded in tension (Figure 11.1). Because of double symmetry, we can use one-quarter for the analysis with symmetry conditions on the edges $x = 0$ and $y = 0$ as shown in Figure 15.17. The stress intensity factor will be determined by the J-integral method. In nondimensional units: $E = 1$, $\nu = 0.3$, $S = 1$. The GUI is not available for the J-integral. The commands must be entered manually on the Command Line.

a. Activate ANSYS and set job name and preferences.
See example 15.3.

b. Establish element type and material properties.

PREPROCESSOR > ELEMENT TYPE > ADD
C ADD
C SOLID
C QUAD 8node 183
C OK
C CLOSE
C PREPROCESSOR > MATERIAL PROP > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1.0 for EX parameter
C in PRXY box
T 0.3 for PRXY parameter
C OK
C MATERIAL > EXIT

c. Establish geometry and mesh the object.

The upper right quarter of the cracked sheet is used. The origin as a KEYPOINT is placed at the crack tip by using two rectangles and then combining them into one material body. The origin must be at the crack tip.

C PREPROCESSOR > MODELING > CREATE > AREAS > RECTANGLE > BY DIMENSIONS
T −2 0 for X1,X2 and 0 10 for Y1,Y2 [note the minus sign]
C APPLY and the first rectangle appears
T 0 8 for X1,X2 and 0 10 for Y1,Y2
C OK and the second rectangle appears
C PLOT CTRLS > NUMBERING
C box after Keypoint Numbers to turn them ON
C box after Line Numbers to turn them ON
C box after Area Numbers to turn them ON
C box after Node numbers to turn them ON
C OK (see all features numbered)
C PREPROCESSOR > MODELING > OPERATE > BOOLEANS > ADD > AREAS
C PICK ALL (this combines the two areas into one area A3)
C PREPROCESSOR > MESHING > SIZE CONTROLS
C CONCENTRATE KPs > CREATE
PICK crack tip at the origin
C OK (menu appears)
Enter .25 for radius of first row of elements
Enter 1 for ratio of second row to first
Enter 6 for no. of elements around circum.
Select SKEWED 1/4 PT from menu for KCTIP
OK
C PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [close warning message]

d. Apply boundary conditions.

C PLOT > LINES
C LOADS > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT >
ON LINES
C bottom right edge of the model (L9)
C APPLY
C UY
Enter 0 for the value of the displacement components
C APPLY
C left edge of the model (L4)
C OK
C UX
OK
C APPLY > STRUCTURAL > PRESSURE > ON LINES
C top edge of *both* of the original two elements (L3 and L10)
C OK
Enter −1 for Load Pressure value [negative for tension]
C OK

e. Create *J*-integral parameters.

C PLOT > ELEMENTS [Utility menu]
C SECLECT > ENTITIES [nodes by num/pick selected]
C OK
Pick node at origin [zoom in to be sure you pick the origin]
C OK
C SELECT > COMP/ASSEMBLY > CREATE COMPONENT
Enter CRACK1 for Cname
C OK
C SELECT > EVERYTHING
[type the following commands on the command line followed by ENTER]
OUTRES,CINT,ALL [output *J*-integral results]
CINT,NEW,1 [new contour integral with ID = 1]
CINT,NAME,CRACK1 [node for path 1]
CINT,NCON,3 [do for three contours]
CINT,SYMM,ON [symmetric about crack line]
CINT,NORM,0,2 [crack normal is *y*-axis]
CINT,LIST [lists parameters in Output Window]

f. Solve equations.

C SOLUTION > SOLV CURRENT LS
C CLOSE on information window
C OK in SOLVE window
C YES in warning window
C CLOSE on information window that solution is complete

g. Calculate the stress intensity factor by the *J*-integral.
GENERAL POSTPROCESSOR
Type PRCINT,1 [on the command line followed by ENTER]
The value of the *J*-integral for three paths is displayed:

$$J_1 = 7.0266,\ J_2 = 6.9961,\ J_3 = 6.9896.$$

The average of J_2 and J_3 is 6.9928, giving $\text{KI} = \sqrt{6.9928} = 2.644$.

15.21 STRETCHING OF A NONLINEAR ELASTIC SHEET

The analysis of elastic materials with large strains is introduced.

This is a solution of stretching of a sheet (Figure 15.18) of a Blatz–Ko material ($\mu = 200$) using a four-node plane stress element (Plane182). We will use the upper half of the sheet and symmetry conditions. Note that $p = 100$ is a given load per unit reference area and not a pressure per unit deformed area.

a. Activate ANSYS and set job name and analysis type to structural.
See example 15.3.
b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
HYPERELASTIC
2D 4 NODE 182 [this is PLANE182 element]
OK
CLOSE
PREPROCESSOR > MATERIAL PROP > MATERIAL MODEL
STRUCTURAL
NONLINEAR
ELASTIC

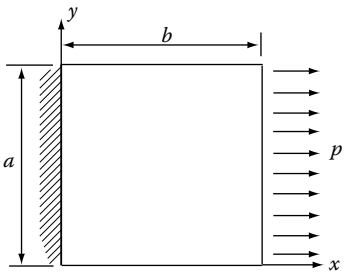


FIGURE 15.18 Stretching of a thin sheet.

HYPERELASTIC
BLATZ–KO (FOAM)
Set mu = 200 [this is the shear modulus, the only parameter in the Blatz–Ko model]
OK
MATERIAL > EXIT

c. Create geometry.
PREPROCESSOR > MODELING > CREATE
AREAS > RECTANGLE > BY DIMENSIONS
T 0 1 0 0.5 for X1,X2,Y1,Y2
OK and the region appears 1 unit wide by 0.5 unit high

d. Mesh the body.
PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MESHING > SIZE CONTROLS
MANUAL SIZE > GLOBAL > SIZE
C box for SIZE
T 0.05 in SIZE box to get elements with side 0.05 unit
OK
PREPROCESSOR > MESHING > MESH > AREAS > FREE
C PICK ALL [20 × 10 mesh appears, node 2 at the lower right corner]

e. Apply boundary conditions.
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT >
ON LINES
PICK BOTTOM EDGE
APPLY
SELECT UY
C on box for VALUES
T 0 for the value of the displacement components
APPLY
PICK LEFT EDGE
OK
SELECT ALL DOF
T 0 for the value of the displacement components
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > FORCE > ON NODES
PICK CORNER NODES ON THE RIGHT SIDE
APPLY
SET FX = 2.5 [corner nodes have half the load]
APPLY
BOX THE REMAINING NODES ON THE RIGHT SIDE
OK

SET FX = 5
OK [Close Warning]

f. Solve for displacements and stresses.

The total load is specified, then reached by ramping up in small time steps. Results are saved after each time step.
SOLUTION > UNABRIDGED MENU
SOLUTION > ANALYSIS TYPE > ANALYSIS OPTIONS
Set NLGEOM ON [this is for $\mathbf{K}_0$]
Set Stress Stiff ON (SSTIF at the bottom) [this is for $\mathbf{K}_1$]
OK
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE
File write frequency: select "every substep"
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME/TIME STEP
Time at end of load step: enter 1
Time step size: enter 0.1
Select RAMPED
Automatic time step: ON
Minimum time step: enter 0.05
Maximum time step: enter 0.1
OK
SOLUTION > SOLV > CURRENT LS
CLOSE information window
OK in solve window
(Convergence information is displayed as solution is marched out.)
CLOSE information window when solution is done

g. Display results: list displacements of selected nodes after each time step.

GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
DEF & UNDEF
OK and see deformed shape
GENERAL POST PROC > LIST RESULTS
NODAL SOLUTION
DOF SOLUTION > X-Component
OK [you should have UX = 0.4339 at node 2]
CLOSE
GENERAL POST PROC > LIST RESULTS
NODAL SOLUTION
STRESS > X-COMPONENT
OK [you should have SX = 120.19 at node 2; this is the true stress T_{11}]
CLOSE

TIME HIST POSTPROC
Close window
DEFINE VARIABLES
Add
Select Nodal DOF Result
OK
T 2 in the box for "list of items" [this is another way of picking the node]
OK
Enter name: UX
Select Translation UX
OK
CLOSE
TIME HIST POSTPROC > MATH OPERATIONS > MULTIPLY
IR = 3 [new variable]
FACTA = 100
IA = 1 [time]
NAME = LOAD [LOAD is 100 × TIME]
OK
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 3
For 2nd variable to list: enter 2
OK [list LOAD and UX for each time step]
CLOSE

h. Plot results: plot the displacement at each time.

TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable: enter 2
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 3
OK
Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: UX
Enter y-axis label: LOAD
SELECT: SPECIFIED Y-RANGE
ENTER Y-RANGE OF 0 TO 110
ENTER NDIV Y-AXIS = 11
OK
Utility Menu: PLOT > REPLOT
The graph can be printed after adjusting colors.

15.22 NONLINEAR ELASTICITY: TENSILE TEST

This is a solution of a tensile test of a unit cube (Figure 15.19) of the Blatz–Ko material (μ = 100) using one 8-node brick and the updated Lagrangian method. One

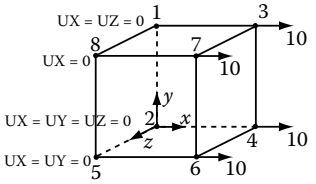


FIGURE 15.19 3D Tensile test.

element is an exact model of the tensile test. The axial load is ramped up to $P_{11} = 40$. Minimum supports are imposed to prevent rigid displacement.

a. Activate ANSYS and set analysis type to structural.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
HYPERELASTIC
C 3D 8 NODE 185 [this is SOLID185 element]
OK
CLOSE
PREPROCESSOR > MATERIAL PROP > MATERIAL MODEL
STRUCTURAL
NONLINEAR
ELASTIC
HYPERELASTIC
BLATZ–KO (FOAM)
Set mu = 100 [this is the shear modulus, the only parameter in the Blatz–Ko model]
OK
MATERIAL > EXIT

c. Create figure.
PREPROCESSOR > MODELING > CREATE > VOLUMES > BLOCK
BY DIMENSIONS
T 0 1 0 1 0 1 for X1,X2,Y1,Y2, Z1,Z2
C OK and the region appears showing the *x*–*y* plane

d. Mesh body.
PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
C OK
PREPROCESSOR > MESHING > MESH TOOL
C SIZE CONTROLS: GLOBL > SET
C box for NDIV
T 1 in NDIV box [one element is exact for the uniform extension]
C OK

C SHAPE: HEX [this forces a brick instead of the default tetrahedron]
C MESH
C PICK ALL [node numbers appear]
C CLOSE [Mesh Tool]

e. Apply boundary conditions.

First, rotate the element so that you can see the base ($x = 0$) with nodes 1-2-5-8. This can be done in the dynamic model mode, which is interactive: on the right-hand side is a menu with 3D pictures of a block. Click on the button at the end of the list that shows two elements (Dynamic Model Mode). The cursor changes to a rotation symbol. Hold down the right mouse button to rotate the element interactively until you can see the face 1-2-5-8.

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT >
ON NODES
C on node 2 (at the origin)
APPLY
C ALL DOF
C on box for VALUES
T 0 for the value of the displacement components
APPLY
REPEAT THIS PROCESS TO SET UX AND UZ TO ZERO AT NODE 1
REPEAT THIS PROCESS TO SET UX AND UY TO ZERO AT NODE 5
REPEAT THIS PROCESS TO SET UX TO ZERO AT NODE 8
[This prevents rigid body motion and keeps the base in the Y–Z plane, but allows contraction.]
ROTATE THE ELEMENT so that you can see the face $x = 1$ with nodes 3-4-6-7
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > FORCE > ON NODES
C ON EACH NODE 3-4-6-7
OK
SELECT FX
T 10 for the VALUE of FX [this fixes the maximum load $P_{11} = 40$]
C OK

f. Solve finite element equations.

SOLUTION > UNABRIDGED MENU
SOLUTION > ANALYSIS TYPE > ANALYSIS OPTIONS
Set NLGEOM ON [this is for $\mathbf{K}_0$]
Set Stress Stiff ON (SSTIF at the bottom) [this is for $\mathbf{K}_1$]
OK
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE
File write frequency: select "every substep"
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME/TIME STEP

Time at end of load step: enter 1
Time step size: enter 0.1
Select RAMPED
Automatic time step: ON
Minimum time step: enter 0.05
Maximum time step: enter 0.1
OK
SOLUTION > SOLV CURRENT LS
CLOSE information window
OK in solve window
(Convergence information is displayed as solution is marched out.)
CLOSE information window when solution is done

g. Display results: list displacements of selected nodes after each time step.

GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
DEF & UNDEF
OK and see deformed shape
GENERAL POST PROC > LIST RESULTS
NODAL SOLUTION
DOF SOLUTION > X-Component
OK [you should have UX = 0.27104 at node 4]
CLOSE
GENERAL POST PROC > LIST RESULTS
NODAL SOLUTION
STRESS > X-COMPONENT
OK [you should have SX = 45.096; this is the true stress T_{11}]
CLOSE
TIME HIST POSTPROC
CLOSE window
DEFINE VARIABLES
ADD
Select Nodal DOF Result
OK
T 4 in the box for "list of items" [this is another way of picking the node]
OK
Enter name: UX
Select Translation UX
OK
ADD
ELEMENT RESULTS
OK
T 1 for the list of items [this is the element number]
OK
T 4 for the list of items [this is the node number for stress output]

OK
Enter name SX
Select Stress Component SX
OK
CLOSE [variable 2 is UX and variable 3 is SX]
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 3
For 2nd variable to list: enter 2
OK [lists SX and UX for each time step]
CLOSE

h. Plot results: plot the displacement at each time.
TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable: enter 2
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 3
OK
Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: UX
Enter y-axis label: SX
SELECT: SPECIFIED Y-RANGE
ENTER Y-RANGE OF 0 TO 50
SELECT: SPECIFIED X-RANGE
ENTER Y-RANGE OF 0 TO 0.3
OK
Utility Menu: PLOT > REPLOT
This graph can be printed for a report after adjusting colors

15.23 COLUMN BUCKLING

This example introduces the FILL command to generate intermediate nodes, the COPY ELEMENT command to proliferate elements, and the PRESTRESS command to save the pre-buckling stress.

A column with one end fixed and the other free has an axial load P as shown in Figure 15.20. The cross section is 0.5 in.^{2}: $A = 0.25$, $I = 1/192$, $L = 100$ in., $E = 30 \times 10^6$ lb/in.^{2} The value of the buckling load is to be calculated. Ten elements of equal length are used.

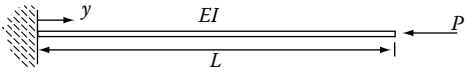


FIGURE 15.20 Column buckling.

a. Activate ANSYS and set preferences.
See example 15.3.

b. Choose elements and materials.
PREPROCESSOR
enter ET,1,BEAM3 on command line [BEAM3 Element]
enter R,1,0.25,1/192,0.5 on command line for A, IZZ, and h
PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
Enter 30e6 for EX parameter
C in NPRXY box
Enter 0.3 for PRXY parameter [PR, denoting Poisson ratio, is actually not used]
OK
MATERIAL > EXIT

c. Create nodes by filling a line and create elements by copying a first element.
PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MODELING > CREATE > NODES > IN ACTIVE CS
Enter Node 1 at 0,0,0
APPLY
Enter node 11 at 0, 100, 0
OK
PREPROCESSOR > MODELING > CREATE > NODES > FILL BETWEEN NODES
C on node 1 then 11
OK
OK in fill window
PREPROCESSOR > MODELING > CREATE > ELEMENTS
AUTONUMBERED > THRU NODES
C on node 1 then 2
OK
PREPROCESSOR > MODELING > COPY > ELEMENTS > AUTONUMBERED
C Pick All
Set ITIME = 10 total copies
OK [a line divided into 10 elements should appear]

d. First, solve for the stress due to a nominal applied load.
SOLUTION > UNABRIDGED MENU
SOLUTION > ANALYSIS TYPE

ANALYSIS OPTIONS > PRESTRESS ON [near the bottom]
OK [this saves the stress state for the eigenvalue calculation]
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON NODES
C node 1
OK
C ALL DOF
C on box for VALUES
T 0 for the value of the displacements and rotation
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
FORCE > ON NODES
C node 11
OK
C FY
C on VALUE box
T value for FY = −1 [negative for compressive force]
OK [force shown on the node]
SOLUTION > SOLVE > CURRENT LS
CLOSE information window [check: Prestress Effects Calculated is YES]
C OK in solve window
CLOSE information window [solution is done]

e. Solve for the multiple of the applied load that produces a zero determinant for the total stiffness matrix.

SOLUTION > ANALYSIS TYPE > NEW ANALYSIS
Select EIGEN BUCKLING
OK [Close Warning If Any]
SOLUTION > ANALYSIS OPTIONS
Enter 1 mode to be extracted [NMODE]
OK in options window
SOLUTION > SOLVE > CURRENT LS
CLOSE information window
OK in solve window
CLOSE information window

f. Display results.

GENERAL POST PROCESSOR > RESULTS SUMMARY
[TIME/FREQ is the factor multiplying the nominal load to get the buckling load, 38.553]
CLOSE
C GENERAL POST PROC > READ RESULTS > FIRST SET
PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see buckled mode shape

15.24 COLUMN POST-BUCKLING

A column with one end fixed and the other free has an axial load P as shown in Figure 15.20. The cross section is 0.5 in.^{2}: $A = 0.25$, $I = 1/192$, $L = 100$ in., $E = 30 \times 10^6$ lb/in.^{2} The value of the buckling load has been calculated as $P_{cr} = 38.553$ lb. The post-buckling displacement is to be determined if the load is increased to $P = 44$. The column is placed so that the top is 1 in. out of alignment in order to simulate an imperfection.

a. Activate ANSYS, change job name and set preferences.
See example 15.3.

b. Choose elements and materials.
See section 15.23.

c. Create nodes by filling a line and create elements by copying a first element.
PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MODELING > CREATE > NODES > IN ACTIVE CS
Enter Node 1 at 0,0,0
APPLY
Enter node 11 at 1, 100, 0 [note $x = 1$ off-set]
OK
FILL BETWEEN NODES
C on node 1 then 11
OK
OK in fill window
PREPROCESSOR > MODELING > CREATE > ELEMENTS
AUTONUMBERED > THRU NODES
C on nodes 1 then 2
OK
PREPROCESSOR > MODELING > COPY > ELEMENTS > AUTONUMBERED
C Pick All
Set ITIME = 10 total copies
OK

d. Apply boundary conditions.
PREPROCESSOR > LOADS > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON NODES
C node 1
OK
C ALL DOF
C on box for VALUES
T 0 for the value of the displacements and rotation
OK

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
FORCE > ON NODES
Select NODE 11
OK
Select FY
C on VALUE box
T value for FY = −44 [negative for compressive force]
OK [force shown on the node]

e. Set solution parameters and solve equations.

SOLUTION > UNABRIDGED MENU
ANALYSIS TYPE > ANALYSIS OPTIONS
Set NLGEOM = ON
Set SSTIFF = ON [drop menu near the bottom]
OK
LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE > EVERY SUBSTEP
OK
TIME/FREQUENC > TIME & SUBSTEPS
TIME = 1
NSUBST = 100
MAXIMUM NO. OF SUBSTEPS = 100
MINIMUM NO. OF SUBSTEPS = 100
OK
SOLVE > CURRENT LS [convergence information displayed]
CLOSE information window
OK
CLOSE information window [solution is done]

f. Display results.

PLOT RESULTS > DEFORMED SHAPE
OK and see post-buckled shape
TIMEHIST PROSTPRO [CLOSE WINDOW]
DEFINE VARIABLES > ADD [nodal DOF selected]
OK
C NODE 11
OK
NAME = UX [translation UX selected]
OK [UX is variable 2]
ADD
REACTION FORCES
OK
C NODE 1
OK
NAME = FY
STRUCTURAL FORCE FY
OK [variable 3 is FY]
CLOSE

SETTINGS > GRAPH
XVAR = SINGLE VARIABLE
SINGLE VARIABLE = 2 [plot UX on *x*-axis]
OK
GRAPH VARIABLES
NVAR1 = 3 [plot FY on *y*-axis]
OK
PLOT CTRLS > STYLE > COLORS
C REVERSE VIDEO
PLOT CTRLS > STYLE > COLORS > GRAPH COLORS
For Graph Curve Number 1 Select Black From Drop Down Menu
OK
PLOT CTRLS > HARD COPY [if desired]
[Choose parameters and either print or save]

15.25 SNAP-THROUGH

This analysis uses the arc-length method to determine intermediate (unstable) equilibrium configurations. Use of the Solution Control menu is introduced.

Under sufficient load, the linkage shown in Figure 15.21 will snap through to another equilibrium position. The exact analysis is discussed in Section 32.1. The initial dimensions are $H = 50$, $V = 1$. The area of the linkage rods is $A = 1$ and the modulus is $E = 10 \times 10^6$. The maximum load is $F = 75$. One-half of the mechanism is used by symmetry and the system is turned upside down so that the displacements are upward (positive *y*-direction).

a. Activate ANSYS and set analysis type.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
C ADD
C LINK
C 3D finit str 180 (Link element)
OK
CLOSE
PREPROCESSOR > REAL CONSTANTS > ADD
C ADD
C OK

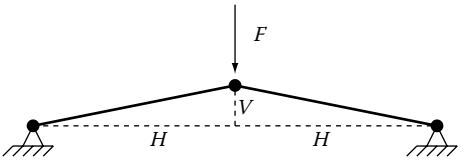


FIGURE 15.21 Linkage snap-through.

C in the AREA box
T 1.0 for area
C OK
C CLOSE
PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL
C LINEAR
C ELASTIC
C ISOTROPIC
T 1e7 for EX parameter
OK (PRXY is not used)
MATERIAL > EXIT

c. Create a line and mesh it as one element.

PREPROCESSOR > MODELING > CREATE > KEYPOINTS > IN ACTIVE CS
NPT = 1
X,Y = 0,0
C APPLY
NPT = 2
X,Y = 50, –1
OK
PREPROCESSOR > MODELING > CREATE > LINES > LINES > STRAIGHT LINE
PICK KP1
PICK KP2
OK
PLOT CTRLS > NUMBERING [top line menu]
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MESHING > SIZE CONTROLS
MANUAL SIZE > GLOBAL > SIZE
C box for NDIV
T 1 in NDIV box to mesh with one truss element
OK
PREPROCESSOR > MESHING > MESH > LINES
C PICK ALL [element appears with node and element numbers]

d. Apply boundary conditions.

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON NODES
PICK NODE 1
OK
C ALL DOF
C on box for VALUES
T 0 for the value of the displacement component
OK

SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON NODES
PICK NODE 2
OK
Select UX [be sure that only UX is highlighted]
T 0 for the value of the displacement component [symmetry condition]
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > FORCE > ON NODES
PICK NODE 2
OK
SELECT FY
VALUE = 75 [this will be the maximum load]
OK

e. Solve for displacements and stresses.

The total load is specified and applied in one step. Results are saved after each substep.
SOLUTION > ANALYSIS TYPE > SOL'N CONTROLS > BASIC
Select Analysis Option: Large Displacement Static [NLGEOM ON]
Number of Substeps selected
Enter Number of substeps = 50
Select Frequency: Write Every Substep
C ADVANCED NONLINEAR
C Activate Arc-Length Method
T Max Multiplier = +1
T Min Multiplier = −1
OK
SOLUTION > SOLVE > CURRENT LS
CLOSE blue information window
C OK in information window ["time" is actually the load multiplier]
CLOSE information window when solution is done

f. Display results: list displacements of selected nodes after each time step.

GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK and see deformed shape
TIME HIST POSTPROC [close information window]
DEFINE VARIABLES [variable 1 labeled time is actually the load multiplier]
Add
Select Nodal DOF Result
OK

Pick node 2 [move window to see model]
OK
Enter name: DISPLACEMENT
Select Translation UY [variable 2 is set to UY]
OK
Close information window
TIME HIST POSTPROC > LIST VARIABLES [Time = Load Multiplier]
For NVAR2 enter 2 [displacement]
OK
CLOSE

g. Plot results: plot the displacement at each time.
Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: DISPLACEMENT
Enter y-axis label: LOAD FACTOR
SELECT SPECIFIED X-RANGE
ENTER X-RANGE OF 0 TO 2.5
SELECT: SPECIFIED Y-RANGE
ENTER Y-RANGE OF –1 TO +1
OK
TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable: enter 2 [displacement on *x*-axis]
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 1 [load multiplier on *y*-axis]
OK

15.26 PLATE BENDING EXAMPLE

The Element Table is used to extract the moments and shears.

This example is a solution of bending of a square plate by a uniform load. Two opposite sides are simply supported with zero twisting moment and the other two opposite sides are simply supported with zero twisting rotation. One-quarter of the plate is modeled with symmetry conditions (Figure 15.22). In nondimensional units: $a = b = 10$, $h = 1$, $\nu = 0.3$, $D = 1$, so that $E = 10.92$. This is a thick plate since $a/h = 10$.

a. Activate ANSYS and set job name and analysis type structural.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
SHELL

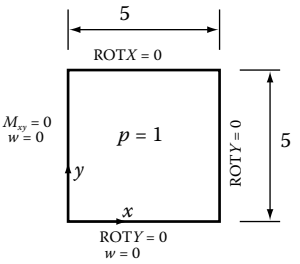


FIGURE 15.22 One-quarter of plate with symmetry.

Plastic 4 node 43 [SHELL43 element]
OK
CLOSE
C REAL CONSTANTS > ADD
C ADD
C OK
ENTER 1 for TK(I) [thickness]
OK
CLOSE
MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
Enter 10.92 for EX parameter [$E = 12(1 - \nu^2)$]
Enter 0.3 for PRXY parameter
OK
MATERIAL > EXIT [menu in upper left corner of window]

c. Create rectangle.
MODELING > CREATE
AREAS > RECTANGLE > BY DIMENSIONS
Enter 0 5 0 5 for X1,X2,Y1,Y2 [$a = b = 10$]
OK and the region appears

d. Mesh region.
MESHING > SIZE CNTRLS > MANUAL SIZE > GLOBAL > SIZE
Enter NDIV = 10
OK
MESHING > MESH > AREAS > FREE
C PICK ALL

e. Apply boundary conditions.
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL >
DISPLACEMENT > ON LINES

C bottom AND left edge of the model
APPLY
C UZ
C on box for VALUES
T 0 for the value of the displacement components
APPLY
C bottom AND right edge of the model
APPLY
C ROTY
APPLY
C top edge
OK
C ROTX
OK
DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON AREAS
C PICK ALL
Enter VALUE = 1 [default LKEY = 1]
C OK [uniform load in positive z-direction]

f. Solve for displacements and stresses.
SOLVE > CURRENT LS
OK
CLOSE
CLOSE

g. Display results: list displacements of selected nodes after each time step.
PLOT CTRLS > NUMBERING
C NODE numbers ON
Select Elem/Attrib numbering = Element Numbers
Select /NUM = Numbers only
OK
GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
DOF SOLUTION > Z-COMPONENT OF DISPLACEMENT
OK [w = 44.379 at the center]
CLOSE
ELEMENT TABLE > DEFINE TABLE > ADD
Enter Lab = MX
C item by sequence number
Enter for SMISC, 4
APPLY
Repeat for MY 5, MXY 6, QX 7. and QY 8
OK
CLOSE
LIST ELEM TABLE
SELECT MX, MY, MXY, QX, QY

OK [see list of stress resultants at center of each element]

h. Graph results.

PATH OPERATIONS > DEFINE PATH > BY NODES
Pick nodes along the base Y = 0 from the left to the center
OK
Enter BASE for the name
OK
CLOSE
MAP ONTO PATH
Enter LAB = QY
C Elem Table Item
Enter for ETAB, QY
OK [ANSYS extrapolates to the boundary]
PLOT PATH ITEM>ON GRAPH
Select QY
OK
PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
C Y-axis range specified
Enter YMIN, YMAX as −5 to + 5
OK
PLOT > REPLOT

15.27 CLAMPED PLATE

This is a solution of bending of a square plate by a uniform load. The edges of the plate are clamped (all DOFs are zero). The lateral load is uniformly distributed. Symmetry is not used. In nondimensional units: $a = b = 20$, $h = 1$, $\nu = 0.3$, $D = 100$, so that $E = 1092$. This is a thick plate since $a/h = 20$.

a. Activate ANSYS and set job name and analysis type structural.
See Example 15.3.

b. Establish element type and material properties.

PREPROCESSOR > ELEMENT TYPE > ADD
ADD
SHELL
Elastic 8 node 281 [SHELL281 element]
OK
CLOSE
C REAL CONSTANTS > ADD
C ADD
C OK
ENTER 1 for TK(I) [thickness]
OK
CLOSE

MATERIAL PROP > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
Enter 1092 for EX parameter [$D = 100$]
Enter 0.3 for PRXY parameter
OK
MATERIAL > EXIT

c. Create rectangle.
MODELING > CREATE
AREAS > RECTANGLE > BY DIMENSIONS
Enter 0 20 0 20 for X1,X2,Y1,Y2 [$a = b = 20$]
OK and the region appears

d. Mesh region.
MESHING > SIZE CNTRLS > MANUAL SIZE > GLOBAL > SIZE
Enter NDIV = 20
OK
MESHING > MESH > AREAS > FREE
C PICK ALL

e. Apply boundary conditions.
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON LINES
C all four edges
OK
C ALL DOF
C on box for VALUES
T 0 for the value of the displacement components
OK
DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON AREAS
C PICK ALL
Enter VALUE = 1 [default LKEY = 1]
C OK [uniform load in the positive z-direction]

f. Solve for displacements and stresses.
SOLVE > CURRENT LS
OK
CLOSE
CLOSE

g. View results.
GENERASL POSTPROC
QUERY RESULTS
SUBGRID SOLU
DOF SOLUTION > UZ
Pick center node
OK [see 2.1236, thin plate theory = 2.016]

15.28 GRAVITY LOAD ON A CYLINDRICAL SHELL

In this example, the command Select > Entities is used to select a set of noses. The use of stored parameters instead of directly entered data, cylindrical coordinates as the active coordinate system, and mapped meshing are introduced.

The cylindrical shell (Figure 15.23) is loaded by gravity in the y-direction. The curved shell is approximated by flat plate elements. The ends are supported by flexible membranes and the sides are free. $E = 3 \times 10^6$ lb/in.^{2}, $\nu = 0$, and the specific weight is 0.208 pci, $r = 300$ in., thickness = 3 in., length = 600 in. Double symmetry will be used.

a. Activate ANSYS and set analysis type.
FILE > CHANGE JOBNAME
T a job name [this is the internal job name, e.g., Shell]
OK
FILE > CHANGE TITLE
T a project name [this is the name on reports, e.g., Shell Roof]
OK
PREFERENCES
C box before structural
OK

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
C ADD
C SHELL
C Elastic 4 node 181 [Shell181]
OK

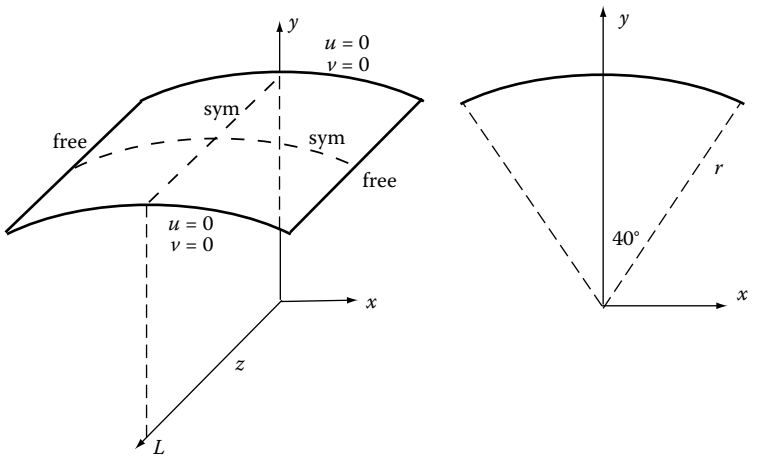


FIGURE 15.23 Cylindrical shell.

CLOSE
PREPROCESSOR > REAL CONSTANTS > ADD
C ADD
C OK
TK(I) = 3
C OK
C CLOSE
PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
EX = 3e6
NUXY = 0.
OK
DENSITY
DENS = .208
OK
MATERIAL > EXIT

c. Enter the value of some parameters for later use.

The name of the parameter and its value or a formula is entered into the selection box.

UTILITY MENU > PARAMETRS > SCALAR PARAMETER
Enter r = 300
ACCEPT
Enter z = 300 [length of one-half of shell]
ACCEPT
CLOSE

d. Change to cylindrical coordinates and create the surface.

We are analyzing one-quarter of the shell and using symmetry conditions.

UTILITY MENU > WORK PLANE > CHANGE ACTIVE CS >
GLOBAL CYLINDRICAL [cylindrical coordinates $X \rightarrow r$, $Y \rightarrow \theta$ (in deg.)]
PREPROCESSOR > MODELING > CREATE > KEYPOINTS > IN ACTIVE CS
NPT = 1
X,Y,Z = *r*,50,0 [the values of parameter *r* will be used]
C APPLY
NPT = 2
X,Y,Z = *r*,90,0
C APPLY
NPT = 3
X,Y,Z = *r*,90,z
C APPLY
NPT = 4
X,Y,Z = *r*,50,z

OK
C ISOMETRIC VIEW [top icon on the right side menu]
PREPROCESSOR > MODELING > CREATE
AREAS > ARBITRARY > THROUGH KPs
PICK KP1
PICK KP2
PICK KP3
PICK KP4
OK [area appears: this is a curved surface in cylindrical coordinates]

e. Mesh the surface.
PREPROCESSOR > MESHING > SIZE CONTROLS
MANUAL SIZE > GLOBAL > SIZE
C box for NDIV
T 20 in NDIV box
OK
PREPROCESSOR > MESHING > MESH > AREAS
MAPPED > 3 OR 4 SIDED
C PICK ALL [20 × 20 mesh appears]

f. Apply boundary conditions using symmetry conditions.
C TOP VIEW [right side menu]
UTILITY MENU > SELECT > ENTITIES
OK [to select nodes, picking menu appears]
C BOX
Select nodes by drawing a box around the nodes on the left edge (global X = 0) with mouse
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT
SYMMETRY B.C. > ON NODES [menu appears]
Norml = Y-axis [actually $\theta = 90°$ plane]
KCN = 1 [global cylindrical coordinate system]
OK [support symbols appear]
UTILITY MENU > SELECT > ENTITIES
OK
C BOX
Select nodes by drawing a box around the nodes on the bottom edge (Z = 300)
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT
SYMMETRY B.C. > ON NODES
Norml = Z-axis
KCN = 1

OK
UTILITY MENU > SELECT > EVERYTHING [i.e., unselect the special set]
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT
ON NODES
C BOX and select nodes on the top edge $Z = 0$
OK
Select UX, UY [membrane support]
Value = 0
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
INERTIA > GRAVITY > GLOBAL
ACELY = 1 [acceleration of gravity in global y-direction, load is $-\rho g$, in negative y-direction or down]
OK
PLOT CTRLS > NUMBERING > NODE NUMBERS ON
ELEM = ELEMENT NUMBERS
/NUM = NUMBERS ONLY
OK
PLOT ELEMENTS [note element nearest center (400) and its nodes]
LIST > ELEMENTS > NODES [note I J K L nodes for the center element, the local x-axis goes from I toward J, and the local y-axis goes from J toward K]
CLOSE

g. Solve for displacement and stress resultants.
SOLUTION > SOLVE > CURRENT LS
OK
CLOSE information windows

h. Display results.
GENERAL POSTPROC > QUERY RESULTS > SUBGRID SOLUTION
DOF SOLUTION > UY [global Y = up]
C node at center of shell (X = 0, Z = 300)
OK
GENERAL POSTPROC > ELEMENT TABLE > DEFINE TABLE
ADD
Enter LAB =N11
Select "By sequence number"
Enter SMISC,1
APPLY
Repeat for N22, 2; N12, 3; M11, 4; M22, 5; M12, 6; Q1, 7; Q2, 8.
OK
CLOSE

GENERAL POSTPROC > ELEMENT TABLE > LIST ELEM TABLE
Select N11, N22, N12, M11, M22, M12Y, Q1, Q2 [values are at the centroid]
OK
CLOSE

i. Plot results.
GENERAL POSTPROC > PATH OPERATIONS > DEFINE PATH > BY NODES
Select node 22 ($z = 300$, $\theta = 90$) and then node 42 ($z = 300$, $\theta = 50$).
OK
Enter path name Z = 300
OK
CLOSE
GENERAL POSTPROC > PATH OPERATIONS > MAP ONTO PATH
Enter LAB = UY [vertical displacement]
Select UY
OK
GENERAL POSTPROC > PATH OPERATIONS > PLOT PATH ITEM > ON GRAPH
Select UY
OK [UY vs. arc length from the top]

15.29 PLATE BUCKLING

A simply supported plate is loaded along one edge by in-plane loads (Figure 15.24). The buckling load is calculated. Use zero rotation about the normal to the edge for the simply supported condition. Use $a = b = 100$, $E = 30 \times 10^6$, $\nu = 0.3$. Use a 20×20 mesh of Shell 281 elements. Use a nominal load of $p = 1000$ and calculate the load factor for buckling.

a. Activate ANSYS and set preferences.
See example 15.3.

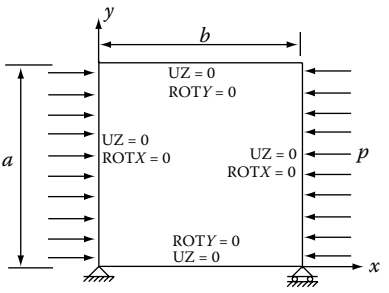


FIGURE 15.24 Plate buckling.

b. Choose elements and materials.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD in the new window
SHELL
Select 8 NODE 281
OK
CLOSE
PREPROCESSOR > REAL CONSTANTS > ADD
ADD
OK
Enter Thickness TK(I) = 1
OK
CLOSE
PREPROCESSOR > MATERIAL PROPS > MATERIAL MODELS
STRUCTURAL
LINEAR
ELASTIC
ISOTROPIC
Enter 30e6 for EX parameter
C in PRXY box
Enter 0.3 for PRXY parameter [the Poisson ratio]
OK
MATERIAL > EXIT

c. Create rectangle.
PREPROCESSOR > MODELING > CREATE > AREAS
RECTANGLE > BY DIMENSIONS
ENTER 0 100 0 100 for X1 X2 Y1 Y2
OK

d. Mesh region.
MESHING > SIZE CNTRLS > MANUAL SIZE > GLOBAL > SIZE
Enter NDIV = 20
OK
MESH > AREAS > FREE
PICK ALL

e. Apply a nominal applied load before buckling.
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON LINES
C all four edges
APPLY
C UZ
C on box for VALUES
T 0 for the value.
APPLY [this is the lateral displacement]
C left and right edges
APPLY
C ROTX [twisting moment restrained]

APPLY
C top and bottom edges
APPLY
C ROTY [twisting moment restrained]
OK
DISPLACEMENT > ON NODES
C ORIGIN
APPLY
C UX AND UY [prevents rigid translation]
APPLY [close warning]
C LOWER RIGHT CORNER [$y = 0$, $x = 20$]
APPLY
SELECT UY ONLY (deselect UX) [prevents rigid rotation]
OK
SOLUTION > DEFINE LOADS > APPLY > STRUCTURAL
PRESSURE > ON LINES
C LEFT AND RIGHT EDGES
OK
C on VALUE box
T value = 1000
OK [close warning]

f. Solve equations and save stress resultants.
SOLUTION > UNABRIDGED MENU
ANALYSIS TYPE > ANALYSIS OPTIONS
PRESTRESS ON [drop menu near the bottom]
OK [this saves the stress state for the eigenvalue calculation]
SOLUTION > SOLVE > CURRENT LS
CLOSE information window [check: Prestress Effects Calculated is YES]
C OK in solve window
CLOSE information window [solution is done]

g. Solve for the multiple of the applied load that produces a zero determinant for the total stiffness matrix.
SOLUTION > ANALYSIS TYPE > NEW ANALYSIS
Select EIGEN BUCKLING
OK [close warning]
SOLUTION > ANALYSIS OPTIONS
Enter 1 mode to be calculated (NMODE)
OK in options window
SOLUTION > SOLVE > CURRENT LS
CLOSE information window
OK in solve window
CLOSE information window

h. Display results.
GENERAL POST PROCESSOR > RESULTS SUMMARY
[TIME/FREQ is the factor multiplying the nominal load to get the buckling load, 10.84]

CLOSE
C GENERAL POST PROC > READ RESULTS > FIRST SET
C GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
C DEF & UNDEF
C OK
Rotate around the x-axis to see buckled mode shape using the Pan–Zoom–Rotate tool or the icons on the right side.

15.30 HEATED RECTANGULAR ROD

A square rod with initial temperature T = 0 is heated on both ends to a temperature T = 1. The nondimensional length is 1 and the nondimensional material properties are $\kappa = 1$, $\rho = 1$, and $c = 1$. The temperature rise at the middle is sought. The four-node thermal element Plane55 is used.

a. Activate ANSYS and set preferences.
FILE > CHANGE JOBNAME
Enter job30 as job name
New Log = YES
OK
PREFERENCES
THERMAL
OK

b. Choose element and material.
PREPROCESSOR
ELEMENT TYPE > ADD
ADD
THERMAL SOLID > QUAD 4NODE 55 [plane55 element]
OK
CLOSE
MATERIAL PROPS > TEMPERATURE UNITS
Select CELSIUS
OK
MATERIAL PROPS > MATERIAL MODELS
THERMAL
CONDUCTIVITY > ISOTROPIC
Enter KXX = 1 [nondimensional parameters]
OK
SPECIFIC HEAT
Enter C = 1
OK
DENSITY
Enter DENS = 1
OK
MATERIAL > EXIT

c. Create geometry.
MODELING > CREATE > AREAS > RECTANGLE > BY DIMENSIONS
Enter X1 = 0, X2 = 1, Y1 = 0, Y2 = 0.1
OK

d. Mesh region.
MESHING > SIZE CNTRLS > MANUAL SIZE > GLOBAL > SIZE
Enter SIZE = 0.05
OK
MESH > AREAS > FREE
PICK ALL [2 × 20 grid of elements]

e. Apply boundary and initial conditions.
LOADS > DEFINE LOADS > SETTINGS > UNIFORM TEMP
Enter TUNIF = 0 [initial condition]
OK
APPLY > THERMAL > TEMPERATURE > ON LINES
Pick the LINES at X = 0 and X = 1 [right and left ends]
OK
Select TEMP and enter VALUE = 1
OK
HEAT FLUX > ON LINES
Select lines Y = 0, and Y = 0.1 [top and bottom]
OK
Enter VALI = 0
OK

f. Solve for temperature history.
SOLUTION
ANALYSIS TYPE > NEW ANALYSIS
Select TRANSIENT
OK
OK [to accept FULL]
LOAD STEP OPTS
OUTPUT CTRLS > DB/RESULTS FILE
Select EVERY SUBSTEP
OK
TIME/FREQUENCY > TIME AND SUBSTPS
Enter TIME = 0.5
NSUBST = 50 [small steps to approximate the exponential]
Select STEPPED [ANSYS actually ramps load up in the first step]
MAX STEPS = 50
MIN STEPS = 50
OK
SOLVE > CURRENT LS

Close information window
OK in solve window
Close done window

g. Contour plot of temperature.
GENERAL POSTPROCOR > READ RESULTS > BY TIME/FREQ
Enter TIME = 0.1
OK
PLOT RESULTS > CONTOUR PLOT > NODAL SOLU
DOF SOLUTION > NODAL TEMPERATURE
OK [note that the temperature varies only lengthwise]

h. View temperature history.
PLOT ELEMENTS
TIMEHIST POSTPRO
Close variable information window
DEFINE VARIABLES
ADD
OK [for Nodal DOF result]
PICK NODE AT X = 0.5 [the middle]
OK
Enter NAME = T
OK [note that variable 2 is temp T at selected node]
CLOSE window
LIST VARIABLES
Enter NVAR1 = 2
OK [temperature at each time is listed]
Close window
GRAPH VARIABLES
Enter NVAR1 = 2
OK [graph of T vs. time appears]

15.31 HEATED CYLINDRICAL ROD

An unrestrained cylindrical rod with initial temperature $T = 0$ is heated on one end to a temperature $T = 1$. The nondimensional radius is 0.1, the nondimensional length is 1, and the nondimensional material properties are $E = 1$, $\nu = 0.3$, $\alpha = 1$, $\kappa = 1$, $\rho = 1$, and $c = 1$. Heat loss on the exterior is neglected. The temperature distribution and the deformations are axisymmetric. The four-node thermal element Plane 55 is used for the temperature calculation and the Plane 182 element is then automatically used for the thermal stress.

a. Activate ANSYS and set preferences.
FILE > CHANGE JOBNAME
Enter job31 as the job name
Select New Log = YES
OK
PREFERENCES

STRUCTURAL
THERMAL
OK

b. Choose thermal element and thermal properties.

PREPROCESSOR
ELEMENT TYPE > ADD
ADD
THERMAL SOLID [QUAD 4NODE 55 selected]
OK
OPTIONS
K3 = AXISYMMETRIC [X becomes radial (*r*), Y become longitudinal (*z*)]
OK
CLOSE
MATERIAL PROPS > TEMPERATURE UNITS
Select CELSIUS
OK
MATERIAL PROPS > MATERIAL MODELS
THERMAL
CONDUCTIVITY > ISOTROPIC
Enter KXX = 1 [nondimensional parameters]
OK
SPECIFIC HEAT
Enter C = 1
OK
DENSITY
Enter DENS = 1
OK
MATERIAL > EXIT

c. Create geometry.

MODELING > CREATE > AREAS > RECTANGLE > BY DIMENSIONS
Enter X1 = 0, X2 = 0.1, Y1 = 0, Y2 = 1
OK

d. Mesh region.

MESHING > SIZE CNTRLS > MANUAL SIZE > GLOBAL > SIZE
Enter SIZE = 0.02
OK
MESH > AREAS > FREE
PICK ALL [5 × 50 grid of elements]

e. Apply boundary and initial conditions.

LOADS > DEFINE LOADS > SETTINGS > UNIFORM TEMP
Enter TUNIF = 0 [initial condition]
OK
APPLY > THERMAL > TEMPERATURE > ON LINES
Pick the LINE at Y = 0 [bottom end]
OK

Select TEMP and enter VALUE = 1
OK
HEAT FLUX > ON LINES [side and other end]
Pick line X = 0.1 and Y = 1
OK
Enter VALI = 0
OK

f. Solve for temperature distribution.

SOLUTION
ANALYSIS TYPE > NEW ANALYSIS
TRANSIENT
OK
OK [to accept FULL]
LOAD STEP OPTS
OUTPUT CTRLS > DB/RESULTS FILE
Select EVERY SUBSTEP
OK
SOLUTION PRINTOUT [this saves nodal temperatures in file job31.rth]
Select EVERY SUBSTEP
OK
TIME/FREQUENCY > TIME AND SUBSTPS
Enter TIME = 1, NSUBST = 50
Select STEPPED [ANSYS actually ramps load up in the first step]
OK
SOLVE > CURRENT LS
Close information window
OK in solve window
Close done window

g. View temperature history.

TIMEHIST POSTPRO
Close variable information window
DEFINE VARIABLES
ADD
OK [for Nodal DOF result]
PICK NODE AT X = 0, Y = 1 [i.e., the end point on the centerline of the cylinder]
OK
Enter NAME = T
OK [note that variable 2 is temp T at selected node]
CLOSE window
LIST VARIABLES
Enter NVAR1 = 2
OK [$T = 0.848$ at $t = 1$]

Close window [for the real temperature, multiply by the actual applied temperature T_0]
GRAPH VARIABLES
Enter NVAR1 = 2
OK [graph of T vs. time appears]
[real time = $(\rho cL^2/\kappa)t$]

h. Element and properties for the thermal stress analysis.

PLOT ELEMENTS [Utility menu]
PREPROCESSOR
ELEMENT TYPE > SWITCH ELEM TYPE
Select THERMAL TO STRUCTURAL
OK [ANSYS changes the element type to PLANE 182]
CLOSE warning
ELEMENT TYPE > EDIT [see element type PLANE182]
OPTIONS
K3 = AXISYMMETRIC
OK
CLOSE
MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL > LINEAR ELASTIC > ISOTROPIC
Enter EX = 1
Enter PRXY = 0.3
OK
THERMAL EXPANSION > SECANT COEFFICIENT > ISOTROPIC
Enter ALPX = 1
OK
MATERIAL > EXIT

i. Boundary conditions and temperature distribution for thermal stress analysis.

DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON NODES
Select Node at X = 0, Y = 1
APPLY
Set ALL DOF = 0 [prevent rigid translation]
APPLY [Close Warning]
Select Node at X = 0, Y = 0
OK
Set UX = 0 [deselect ALL DOF]
OK [prevent rigid rotation]
DEFINE LOADS > APPLY > STRUCTURAL > TEMPERATURE FROM THERM ANALY
Enter Time-point = 0.4
Enter Fname = job31.rth [read saved temperature data]
OK

j. Solve for thermal stress.
SOLUTION
ANALYSIS TYPE > NEW ANALYSIS
STATIC
OK
SOLVE > CURRENT LS
Close Status window
OK solve window
Close solution done

k. View Results.
GENERAL POSTPROC
PLOT RESULTS > CONTOUR PLOT > NODAL SOLUTION
STRESS > Y-COMPONENT [thermal stress]
OK
LIST RESULTS > NODAL SOLUTION > STRESS > Y-COMPONENT
OK [for the real stress, multiply by $E\alpha T_0$]
CLOSE

15.32 HEATED DISK

An unrestrained hollow cylinder rod with an internal radius of 5 in. and an outer radius of 10 in. is heated to a steady state. The temperature on the inner radius is 480°F and 75°F on the outer radius. Material properties: $E = 2.9 \times 10^7$ lb/in., $\nu = 0.3$, $\rho = 0.284$ lbm/in.^{3}, $\alpha = 6.67 \times 10^{-6}$ 1/°F, $\kappa = 8.092\ 10^{-4}$ BTU/s. in. °F, $c = 0.1036$ BTU/lbm. °F. Temperature distribution and thermal stress are to be determined. The four-node thermal element Plane 55 is used for the temperature calculation and the Plane 182 element is then automatically used to calculate the thermal stress. For a very long or very short cylinder, this problem is one-dimensional, but a ¼ cross section is maintained for clarity (Figure 15.25). The body is analyzed as a plane strain.

a. Activate ANSYS and set preferences.
FILE > CHANGE JOBNAME
Enter job32 as job name
Select NEW LOG = YES
OK
PREFERENCES

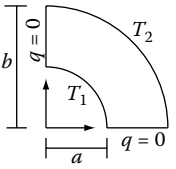


FIGURE 15.25 Heated disk.

STRUCTURAL
THERMAL
OK

b. Choose thermal element and thermal properties.

PREPROCESSOR
ELEMENT TYPE > ADD
ADD
THERMAL SOLID [QUAD 4NODE 55 selected]
OK
CLOSE
MATERIAL PROPS > TEMPERATURE UNITS
Select FAHRENHEIT
OK
MATERIAL PROPS > MATERIAL MODELS
THERMAL
CONDUCTIVITY > ISOTROPIC
Enter KXX = 8.092e – 4
OK
SPECIFIC HEAT
Enter C = 0.1036
OK
DENSITY
Enter DENS = 0.284
OK
MATERIAL > EXIT

c. Create geometry.

MODELING > CREATE > KEYPOINTS > IN ACTIVE CS
Enter 1 at 5, 0
APPLY
Enter 2 at 10, 0
APPLY
Enter 3 at 0, 10
APPLY
Enter 4 at 0, 5
OK
WORKPLANE > CHANGE ACTIVE CS > TO GLOBAL CYLINDRICAL
CREATE > LINES > LINES > IN ACTIVE CS
Pick KP 1 then 2
Pick KP 1 then 4 [arc in cylindrical system]
Pick KP 2 then 3 [arc in cylindrical system]
Pick KP3 then 4
OK
CREATE > AREAS > ARBITRARY > BY LINES
Pick the lines sequentially
OK

d. Mesh region.

MESHING > SIZE CNTRLS > MANUAL > LINES > ALL LINES
Enter NDIV = 20
OK
MESH > AREAS > FREE
PICK ALL [uniform 20 × 20 mesh]

e. Apply boundary conditions on temperature.

LOADS > DEFINE LOADS > APPLY > THERMAL > TEMPERATURE > ON LINES
Pick the inner arc
APPLY
Select TEMP and enter VALUE = 480
APPLY
Pick the outer arc
OK
Enter VALUE = 75
OK
APPLY > THERMAL > HEAT FLUX > ON LINES
Pick the lines X = 0 and Y = 0
OK
VALUI = 0
OK

f. Solve for temperature distribution and save results.

SOLUTION [steady-state is the default]
LOAD STEP OPTS
OUTPUT CTRLS
SOLU PRINTOUT [this saves nodal temperatures in file job32.rth]
Select LAST SUBSTEP
OK
SOLVE > CURRENT LS
Close information window
OK in solve window
Close done window

g. View temperature distribution.

GENERAL POSTPROC > PLOT RESULTS
CONTOUR PLOT > NODAL SOLU
DOF SOLUTION > NODAL TEMPERATURE
OK [Contour plot shows axisymmetric temperature]

h. Change element for stress analysis.

PLOT ELEMENTS [Utility menu]
PREPROCESSOR
ELEMENT TYPE > SWITCH ELEM TYPE
Select THERMAL TO STRUCTURAL

OK [ANSYS changes the element type to PLANE 182]
CLOSE warning
ELEMENT TYPE > EDIT > OPTIONS
K3 = PLANE STRAIN
OK
CLOSE

i. Mechanical properties.
MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL > LINEAR > ELASTIC > ISOTROPIC
Enter EX = 2.9e7
Enter PRXY = 0.3
OK
THERMAL EXPANSION > SECANT COEFFICIENT > ISOTROPIC
Enter ALPX = 6.67e−6
MATERIAL > EXIT

j. Mechanical boundary conditions and thermal load.
SOLUTION
DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON LINES
Select Line Y = 0
APPLY
Set UY = 0 [symmetry]
APPLY [Close Warning]
Select Line X = 0
OK
Set UX = 0 [symmetry]
OK
DEFINE LOADS > APPLY > STRUCTURAL > TEMPERATURE
FROM THERM ANALY
Enter Time-point = 1
Enter Fname = job32.rth [read saved temperature data]
OK

k. Solve for thermal stress.
SOLVE > CURRENT LS
Close Status window
OK solve window
Close solution done

l. View results.
GENERAL POSTPROC
QUERY RESULTS > SUBGRID SOLU
STRESS > SX
OK
Pick node at X = 0, Y = 5 $\tau_{\theta\theta}$ = –70,678 lb/in.^{2}
Pick node at X = 0, Y = 10 $\tau_{\theta\theta}$ = 45,236 lb/in.^{2}
OK

15.33 TRUSS CONTACTING A RIGID FOUNDATION

The truss is forced into contact with a rigid foundation (Figure 15.26) with $L = 10$, $g = 0.3$. Member 1 has $A = 0.1$ and $E = 12{,}500$. Member 2 has $A = 0.2$ and $E = 12{,}500$. There is no resistance to slip after contact. The loads are ramped up to $F_x = 27$ and $F_y = 12.8$. Equation 14.28 shows that the simultaneous loading causes contact at $F_x = 13.94$, $F_y = 6.61$, $U = 0.6495$. This is a node to surface contact. Use Contact Element 175, Target Element 169, and the Lagrange multiplier method with default parameters.

a. Activate ANSYS and select preferences.
See example 15.3.

b. Set up parameters.
PARAMETERS > SCALAR PARAMETERS >
G = 0.3 [gap size]
ACCEPT
MU = 0 [friction factor]
ACCEPT
CLOSE

c. Element types, real constants, material properties.
PREPROCESSOR
ELEMENT TYPE
ADD
ADD
LINK > 3D finit str 180
OK
ADD
CONTACT > 2D TARGET 169
OK
ADD
PT-TO-SURF 175
OK
Select TYPE 3 CONTA175

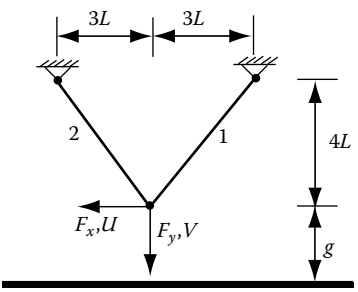


FIGURE 15.26 Truss contact problem.

OPTIONS
K2 = LAGRANGE METHOD
OK
CLOSE
REAL CONSTANTS > ADD
ADD
CONTA175
OK
OK [accept defaults]
ADD
LINK1
OK
AREA = 0.1 [set 2]
OK
ADD
LINK1
OK
AREA = 0.2 [set 3]
OK
CLOSE
MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL > LINEAR > ELASTIC > ISOTROPIC
EX = 12500 [material 1]
PRXY = 0.3 [not used]
OK
C FRICTION COEFFICIENT
MU = 0
OK
MATERIAL > EXIT

d. Create nodes and elements.

MODELING > CREATE
NODES > IN ACTIVE CS
NODE = 1
X,Y,Z = 0,0,0
APPLY
NODE = 2
X,Y,Z = 30,40,0
APPLY
NODE = 3
X,Y,Z = −30,40,0
OK
ELEMENTS > ELEM ATTRIBUTES
TYPE = LINK1
MAT = 1
REAL = 2
OK

AUTO/NUMBERED > THRU NODES
PICK 1 AND 2
OK
ELEMENTS > ELEM ATTRIBUTES
TYPE = LINK1
MAT = 1
REAL = 3
OK
AUTO/NUMBERED > THRU NODES
PICK 1 AND 3
OK
ELEMENTS > ELEM ATTRIBUTES
TYPE = CONTA175
MAT = 1
REAL = 1
OK
SURF / CONTACT
NODE TO SURF
OK
PICK NODE 1
OK
MODELING > CREATE
KEYPOINTS > IN ACTIVE CS
NPT = 10
X,Y,Z = +30,−*g*,0
APPLY
NPT = 11
X,Y,Z = −30,−*g*,0
OK
LINES > LINES > IN ACTIVE CS
PICK 10 THEN 11 IN THAT ORDER [so that the normal will have the proper direction]
OK
MESHING > MESH ATTRIBUTES > DEFAULT ATTRIBS
TYPE = TARGET169
REAL = 1
OK
MESH > LINES
PICK LINE
OK [a target not overlaid is rigid]

e. Apply boundary conditions.

LOADS > DEFINE LOADS > APPLY
STRUCTURAL > DISPLACEMENT > ON NODES
PICK NODES 2 AND 3
OK

ALL DOF
VALUE = 0
OK
FORCE/MOMENT > ON NODES
PICK NODE 1
OK
APPLY
LAB = FX
VALUE = −27 [i.e., to the left]
APPLY
PICK NODE 1
OK
LAB = FY
VALUE = −12.8 [i.e., downward]
OK

f. Solve for displacements.

SOLUTION
UNABRIDGED MENU
LOAD STEP OPTS
OUTPUT CONTROLS > DB RESULTS FILE
SELECT EVERY SUBSTEP
OK
TIME/FREQUENCY > TIME AND SUBSTEPS
TIME = 1
NSUBST = 50
MINIMUM NO. = 50
MAXIMUM NO. = 50
OK
SOLVE > CURRENT LS
CLOSE INFO WINDOW
OK IN SOLVE WINDOW [see graph of convergence]
CLOSE SOLUTION IS DONE WINDOW

g. View results.

GENERAL POSTPROC
READ RESULTS
LAST SET
PLOT RESULTS > DEFORMED SHAPE
DEF + UNDEFORMED [displacement is magnified so that the node appears to penetrate the surface]
OK
LIST RESULTS > NODAL SOLUTION
CONTACT > STATUS
OK [STAT = 2 indicates contact]
CONTACT > PENETRATION [PENE = 0.0000, i.e., no penetration]

CONTACT > PRESSURE [PRES = 12.000]
CLOSE
LIST RESULTS > NODAL SOLUTION
DOF SOLUTION > X-COMPONENT OF DISPLACEMENT
OK [UX = −1.1333]
CLOSE
LIST RESULTS > NODAL SOLUTION
DOF SOLUTION > Y-COMPONENT OF DISPLACEMENT
OK [UY = −0.3, i.e., gap closed]
CLOSE

h. View time history.

TIME HIST POSTPROC
CLOSE INFO WINDOW
DEFINE VARIABLES > ADD > NODAL DOF RESULT
OK
PICK NODE 1
OK
NAME = UX
OK [variable 2 is UX]
ADD > NODAL DOF RESULT
OK
PICK NODE 1
OK
NAME = UY
SELECT UY
OK [variable 3 is UY]
CLOSE
LIST VARIABLES
NVAR1 = 2
NVAR2 = 3
OK [note contact UY = −0.3, 0.629 ≤ UX ≤ −0.653 at ≤0.52 of total load]
CLOSE

15.34 COMPRESSION OF A RUBBER CYLINDER BETWEEN RIGID PLATES

Coupling of nodes is introduced and subregions are used to control meshing.

A long rubber cylinder is compressed between rigid plates (Figures 15.27 and 15.28). The cylinder is analyzed as a plane strain problem with $R = 0.5$ in., $E = 500$ lb/in.^{2} (rubber), $\nu = 0.499$. Since the material is nearly incompressible, the mixed (U–P) formulation is used. Because of the double symmetry, only ¼ of the cylinder is modeled. The plane $y = 0$ is assumed to remain plane and the nodal displacements UY on the plane are coupled. A vertical displacement, $d = 0.05$ in. of the plane is imposed. The required force F and the contact region b are calculated. The expected results are given in Section 14.3.1.

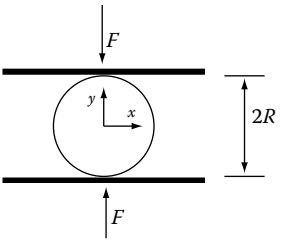


FIGURE 15.27 Compressed cylinder.

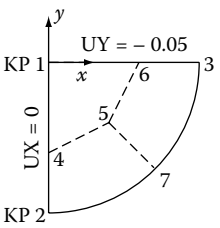


FIGURE 15.28 Subregions for one-quarter.

a. Activate ANSYS and set analysis type to structural.
See example 15.3.

b. Establish element type and material properties.
PREPROCESSOR > ELEMENT TYPE > ADD
C ADD
C SOLID > 4 NODE 182
OK
OPTIONS
SET K3 TO PLANE STRAIN
SET K6 TO MIXED U/P
OK
ADD
CONTACT > 2D TARGET 169 [surface-to-surface contact pair]
OK
ADD
2 ND SURF 171
OK
C TYPE 3 CONTA 171
OPTIONS [accept defaults]
OK
CLOSE [Real Constants > add > add > CONTA171
could be used here to adjust contact parameters]
PREPROCESSOR > MATERIAL PROP > MATERIAL MODEL
C STRUCTURAL > LINEAR > ELASTIC > ISOTOPIC

EX = 500 [PSI]
PRXY = 0.499
OK
MATERIAL > EXIT

c. Create lower right quarter of the cylinder and divide it into areas to control meshing.

PARAMETERS > SCALAR PARAMETERS [Utility menu]
C SELECTION BOX
R = 0.5
ACCEPT
CLOSE
WORK PLANE > CHANGE ACTIVE CS TO > GLOBAL
CYLINDRICAL MODELING > CREATE > KEYPOINTS > IN ACTIVE CS
NPT = 1, X = 0, Y = 0
APPLY
Repeat for keypoints at
2, R, −90
3, R, 0
4, 0.5*R, −90 [some interior points to control meshing]
5, 0.6*R, −45
6, 0.5*R, 0
7, R, −45
OK
MODELING > CREATE > LINES > LINES > IN ACTIVE COORD
PICK 2 AND 7 [cylindrical system for curved lines}
PICK 7 AND 3
OK
WORK PLANE > CHANGE ACTIVE CS TO > GLOBAL CARTESIAN
MODELING > CREATE > AREAS > ARBITRARY > THROUGH KPs
Pick 2, 7, 5, 4
APPLY
Pick 7, 3, 6, 5
APPLY
Pick 4, 5, 6, 1
OK

d. Mesh cylinder.

MESHING > MESH TOOL > MESH [accept default mesh size]
C PICK ALL [10 elements along the arc]
REFINE [on mesh tool]
C PICK ALL
OK [accept default = level 1]
CLOSE mesh tool [20 elements long the arc]

e. Create the rigid base as a straight line tangent to the cylinder.

MODELING > CREATE > KEYPOINTS > IN ACTIVE CS

NPT = 10, X = +2 * R, Y = −R
APPLY
NPT = 11, X = −2 * R, Y = −R
OK
MODELING > CREATE > LINES > LINES > IN ACTIVE COORD
Pick KP 10 then 11 [must pick in this order to get correct normal]
OK

f. Mesh rigid base.
MESHING > MESH ATTRIBUTES > DEFAULT ATTRIBS
TYPE = TARGE169
OK
PLOT > LINES
MESH > LINES
PICK the rigid base [defaults to one element]
OK

g. Overlay contact elements on the surface of the cylinder.
MODELING > CREATE > ELEMENTS > ELEM ATTRIBUTES
TYPE = CONTA171
OK
SURF / CONTACT > SURF TO SURF
OK [accept top surface]
Pick about 10 nodes along the arc surface of the cylinder up from the bottom
OK [note outward normal to the elements on the curved edge as required]

h. Restrict the nodes on the plane of symmetry (Y = 0) to a common UY.
PREPROCESSOR > COUPLING > COUPLE DOFs
Pick nodes along the plane $y = 0$ by box [along the top]
OK
NSET = 1
LAB = UY [makes the UY component the same]
OK

i. Apply boundary conditions.
LOADS > DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON NODES
PICK nodes along center line ($x = 0$) by box
OK
UX = 0 [symmetry condition]
OK
DEFINE LOADS > APPLY > STRUCTURAL > DISPLACEMENT > ON NODES
PICK top right corner node (27) on symmetry line $y = 0$
OK
UY = −0.05 [note negative sign]
OK

j. Solve problem step by step.

SOLUTION > ANALYSIS TYPE > SOL'N CONTROLS
Time at end of load step = 1
Number of substeps = 10
OK
SOLUTION > SOLVE > CURRENT LS
CLOSE information window
OK in solve window
(Convergence information is displayed as solution is marched out.)
CLOSE information window when solution is done

k. Display results.

PLOT CTRLS > STYLE > DISPLACEMENT SCALING
Select 1.0 [no magnification]
OK
PLOT CTRLS > NUMBERING
Node Numbers = ON [zoom in as required]
GENERAL POST PROC > READ RESULTS > LAST SET
GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
OK and see deformed shape [apparent contact by nodes 1-3-4-5]
GENERAL POST PROC > LIST RESULTS
REACTION SOLUTION > FY
OK
Record FY at node 27 [FY = –12.507]
CLOSE
GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
CONTACT > CONTACT STATUS
OK [status = 2 for nodes 1-3-4-5 indicates contact]
[status = 1 for node 6 indicates near contact]
CONTACT > CONTACT PRESSURE [p = 104.67 at node 1]
CLOSE
LIST > NODES [Utility menu]
OK
Record X coordinates for nodes 5 and 6 [X5 = 0.12751, X6 = 0.16705]
CLOSE
GENERAL POST PROC > LIST RESULTS > NODAL SOLUTION
DOF SOLUTION > X COMPONENT
OK
Record UX5 and UX6 [UX5 = 0.00291, UX6 = 0.00629]
[deformed X-position of node 5 is X5 + UX5 = 0.13042]
[deformed X-position of node 6 is X6 + UX6 = 0.17334]
CLOSE [contact boundary at $0.13 < x < 0.17$]

15.35 HERTZ CONTACT PROBLEM

Use of local coordinate systems is introduced. A contact pair is created using the Contact Wizard of the ANSYS Contact Manager.

The contact between two long cylinders is analyzed by assuming double symmetry (Figure 15.29). The following sample instructions are for R_1 = 10 mm, R_2 = 13 mm, $\nu_1 = \nu_2 = 0.25$, $E_1 = E_2 = 30{,}000$ N/mm^2. The angle defining the potential contact region is $\alpha_1 = \alpha_2 = 8°$. Apply a displacement $d = 0.1$ and determine the total force, the maximum contact pressure, and the contact region. The theory is discussed in Section 14.3.2.

a. Activate ANSYS and set preferences.
See example 15.3.

b. Element types, real constants, material properties.
PREPROCESSOR
ELEMENT TYPE
ADD
ADD
SOLID
4 NODE 182 [PLANE 182 element]
OK
OPTIONS
Set K3 to PLANE STRAIN
OK
CLOSE
MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL > LINEAR > ELASTIC > ISOTROPIC

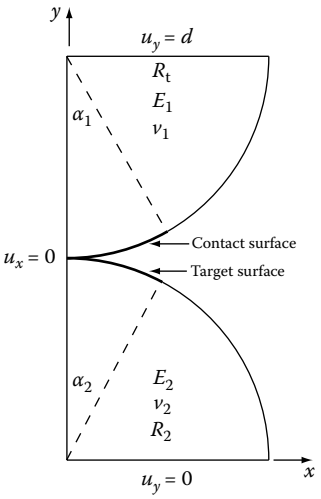


FIGURE 15.29 Contact between cylinders.

EX = 30000
PRXY = 0.25
OK
MATERIAL > EXIT

c. Create the target (large cylinder) and mesh it.

WORKPLANE [Utility menu]
CHANGE ACTIVE CS TO > GLOBAL CYLINDRICAL
MODELING > CREATE
KEYPOINTS > IN ACTIVE CS [for large cylinder]
NPT = 1
X,Y,Z = 0,0,0
APPLY
NPT = 2
X,Y,Z = 13,0,0
APPLY
NPT = 3
X,Y,Z = 13,82,0
APPLY
NPT = 4
X,Y,Z = 13,90,0
APPLY
NPT = 5
X,Y,Z = 11,90,0
OK
LINES > LINES > IN ACTIVE CS
PICK 1 AND 5
PICK 1 AND 2
PICK 2 AND 3 [circular arc]
PICK 3 AND 4
PICK 4 AND 5
OK
WORKPLANE > LOCAL COORDINATE SYSTEMS
CREATE LOCAL CS > AT SPECIFIED LOC
PICK KP 4 (X,Y,Z) = 0,13,0
OK
KCS = CYLINDRICAL
OK [local cylindrical coordinate system 11 at KP 4]
LINES > IN ACTIVE CS
PICK 3 AND 5 [csys 11 necessary to get concave arc]
OK
WORKPLANE [Utility menu]
CHANGE ACTIVE CS TO > GLOBAL CYLINDRICAL
AREAS > ARBITRARY > THROUGH KPs
PICK 1,2,3,5
APPLY

PICK 5, 3, 4
OK
MESHING > MESH TOOL
SIZE CONTROLS GLOBL SET
NDIV = 10
OK
MESH [in Mesh tool]
PICK ALL
CLOSE [mesh tool]
PLOT CTRLS > NUMBERING > NODE NUMBERS ON
OK

d. Create small cylinder and mesh it.

WORKPLANE > CHANGE ACTIVE CS TO > GLOBAL CARTESIAN
WORKPLANE > LOCAL COORDINATE SYSTEMS
CREATE LOCAL CS > AT SPECIFIED LOC
Enter coordinates 0, 23, 0 [center of small cyl]
OK
KCN = 12 [new active CS]
THXY = −90 [rotate X-axis downward]
OK
MODELING > CREATE
KEYPOINTS > IN ACTIVE CS [for small cylinder]
NPT = 11
X,Y,Z = 0,0,0
APPLY
NPT = 12
X,Y,Z = 10,0,0
APPLY
NPT = 13
X,Y,Z = 10,8,0
APPLY
NPT = 14
X,Y,Z = 10,90,0
APPLY
NPT = 15
X,Y,Z = 8,0,0
OK
LINES > LINES > IN ACTIVE CS
Enter 12,13 [on picking menu]
APPLY
Enter 13,14 [circular arc]
APPLY
Enter 14,11
APPLY
Enter 11,15
APPLY

Enter 15,12
OK
WORKPLANE > CHANGE ACTIVE CS TO > SPECIFIED COORD SYS
KCN = 11
OK
LINES > IN ACTIVE CS
Enter 13,15 [csys 11 necessary to get arc concave]
OK
WORKPLANE > CHANGE ACTIVE CS TO > SPECIFIED COORD SYS
KCN = 12 [back to local coords for small cyl]
OK
AREAS > ARBITRARY > THROUGH KPs
Enter 12,13,15
APPLY
Enter 15,13,14,11
OK
MESHING > MESH TOOL
SIZE CONTROLS GLOBL SET
NDIV = 10
OK
MESH
PICK AREAS 3 AND 4 [small cylinder]
OK [zoom in and note nodes on target surface: 202 to 212]
CLOSE [mesh tool]

e. Create contact surfaces.

PLOT CTRLS > NUMBERING
LINE NUMBERS ON
PLOT > LINES
ZOOM IN ON CONTACT REGION
MODELING > CREATE > CONTACT PAIR [contact manager]
WIZARD (LEFT MOST ICON) [lines and flexibility selected]
PICK TARGET
SELECT CONTACT LINE ON LOWER CYLINDER [L4]
OK
NEXT [lines and surface-to-surface selected]
PICK CONTACT
SELECT CONTACT LINE ON UPPER CYLINDER [L7]
OK
NEXT

CREATE [note: normals outward from each cylinder]
FINISH [note description of contact pair]
CLOSE CONTACT MANAGER [C on X]

f. Apply supports.

WORKPLANE > CHANGE ACTIVE CS TO > GLOBAL CARTESIAN
ZOOM TO FIT [logo on right]
COUPLING/CEQN > COUPLE DOFs
PICK NODES ON TOP EDGE BY BOX [of small cylinder]
OK
NSET = 1
LAB = UY
OK
PLOT LINES
LOADS > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON LINES
PICK BOTTOM LINE (Y = 0)
APPLY
UY
VALUE = 0
APPLY
PICK FOUR VERTICAL LINES (X = 0)
OK
UX
VALUE = 0
OK

g. Apply displacement on upper plane.

PLOT > ELEMENTS
LOADS > DEFINE LOADS > APPLY > STRUCTURAL >
DISPLACEMENT > ON NODES
PICK UPPER RIGHT CORNER [node 293]
OK
UY
VALUE = −0.1 [note the minus sign]
OK
SOLUTION
UNABRIDGED MENU
LOAD STEP OPTS
TIME/FREQUENCY > TIME AND SUBSTEPS
TIME = 1
NSUBST = 10
OK

h. Solve for displacement and stress.

SOLUTION > SOLVE > CURRENT LS
CLOSE INFO WINDOW

OK
CLOSE solution is done

i. View results.

PLOT CTRLS > STYLE > DISPLACEMENT SCALING
Select 1.0
OK
GENERAL POSTPROC
PLOT RESULTS > DEFORMED SHAPE
OK
PLOT CTRLS > NODE NUMERS ON
Zoom in to see contact nodes: last one is 206 or 207
LIST RESULTS > REACTION SOLUTION

OK	[FY = –314 at node 293]
CLOSE	
LIST RESULTS > NODAL SOLUTION	
CONTACT > STATUS	[STAT = 2 indicates contact at node 206]
OK	[STAT = 1.5 indicates near contact at node 207]
Note pressure at center node	[731 at node 202]
Note furthest node in contact	[node 206]
Note location of next node	[node 207]
LIST NODES	
OK	
Note x-coordinate of last contact node	[0.41876 at node 206]
Note x-coordinate of next node	[0.55822 at node 207]
CLOSE	
LIST RESULTS > NODAL SOLUTION	
DOF SOLUTION > X COMPONENT	
OK	
Record UX5 and UX6	[UX206 = −0.00534, UX207 = −0.00589] [deformed X-position of node 206 is X + UX = 0.41342] [deformed X-position of node 207 is X + UX = 0.55233]
CLOSE	[contact boundary at $0.413 < x < 0.552$]

15.36 ELASTIC ROD IMPACTING A RIGID WALL

This example demonstrates the use of the Solution Controls menu for transient analysis. The analysis uses node to surface contact, contact element 175, target element 169, using the default Augmented Lagrangian method with increased contact stiffness FKN. The rod (Figure 15.30) is made of steel: $A = 1$ in.^{2}, $E = 2.92 \times 10^7$ lb/in.^{2},

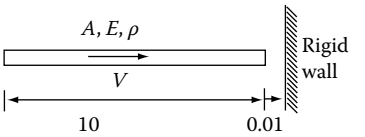


FIGURE 15.30 Rod impacts wall.

$\rho = 0.73 \times 10^{-3}$ lbf-s^2/in.^{4} The rod (Link element) approaches the wall with initial velocity $V = 200$ in./s^2 from a gap of 0.01 in.

a. Activate ANSYS and set preferences.
See example 15.3.

b. Element types, real constants, material properties.
PREPROCESSOR
ELEMENT TYPE
ADD
ADD
LINK > 2D SPAR 1
OK
ADD
CONTACT > 2D TARGET 169
OK
ADD
PT-TO-SURF 175
OK
Select TYPE 3 CONTA175
OPTIONS
OK [accept augmented Lagrangian method]
CLOSE
REAL CONSTANTS > ADD
ADD
LINK180
OK
AREA = 1 [set 1]
OK
ADD
CONTA175
OK [set 2]
FKN = 10 [increase contact stiffness to reduce penetration]
OK
CLOSE
MATERIAL PROPS > MATERIAL MODELS
C STRUCTURAL > LINEAR > ELASTIC > ISOTROPIC
EX = 2.92E7

PRXY = 0.3 [not used]
OK
C DENSITY
DENS = 0.73e−3
OK
MATERIAL > EXIT
PARAMETERS > SCALAR PARAMETERS
G = 0.01 [gap size]
ACCEPT
CLOSE

c. Create line and mesh it.

MODELING > CREATE
KEYPOINTS > IN ACTIVE CS
NPT = 1
X,Y,Z = −10−G,0,0 [end of the rod]
APPLY
NPT = 2
X,Y,Z = −G,0,0 [end of the rod]
OK
MODELING > CREATE > LINES > LINES > IN ACTIVE CS
Pick KP1 AND KP2
OK
MESHING > SIZE CNTRLS > MANUAL SIZE > GLOBAL > SIZE
NDIV = 20
OK
MESH > LINES
PICK ALL
PLOT CTRLS > WINDOW CONTROLS > WINDOW OPTIONS
/TRIAD = AT BOTTOM LEFT
OK [moves axis symbols out of the way]

d. Add contact elements.

MODELING > CREATE > ELEMENTS
ELEM ATTRIBUTES
TYPE = CONTA175
REAL = 2
OK
SURF / CONTACT
NODE TO SURF
OK
PICK NODE ON RIGHT END (2)
OK
MODELING > CREATE > KEYPOINTS > IN ACTIVE CS
NPT = 10
X,Y,Z = 0,2,0

APPLY
NPT = 11
X,Y,Z = 0,–2,0
OK
LINES > LINES > IN ACTIVE CS
PICK 10 THEN 11 IN THAT ORDER [to get the correct normal]
OK
MESHING > MESH ATTRIBUTES > DEFAULT ATTRIBS
TYPE = TARGET169
OK REAL = 2 selected]
MESH > LINES
PICK LINE FROM KP10 TO KP11
OK

e. Apply boundary and initial conditions.
LOADS > DEFINE LOADS > APPLY

STRUCTURAL > DISPLACEMENT > ON NODES
PICK NODES ON THE ROD BY A BOX
OK
UY
VALUE = 0
OK
APPLY > INITIAL CONDIT'N > DEFINE
PICK NODES ON ROD BY BOX
OK
Lab = UX
VALUE2 = 200 [initial velocity]
OK

f. Solve for displacements.
SOLUTION
ANALYSIS TYPE > NEW ANALYSIS > TRANSIENT
OK
LUMPM = YES
OK
SOL'N CONTROLS [Small Displacement Transient selected]
Time at end of loadstep = 2e–4
Number of substeps = 100
Max no. of substeps = 100
Min no. of substeps = 100
Write Items to Results File = All Solution Items
Frequency = Write Every Substep
OK
SOLVE > CURRENT LS
CLOSE INFO WINDOW

OK IN SOLVE WINDOW
CLOSE SOLUTION IS DONE WINDOW

g. View history of displacement.

TIME HIST POSTPROC
CLOSE INFO WINDOW
DEFINE VARIABLES > ADD > NODAL DOF RESULT
OK
PICK RIGHT END NODE 2
OK
NAME = U
OK [variable 2 is UX at node 2]
CLOSE
GRAPH VARIABLES
NVAR1 = 2
OK [print if desired]

15.37 CURVE FIT FOR NONLINEAR ELASTICITY USING BLATZ–KO MODEL

The ANSYS curve fit for hyperelastic materials is used to interpret test data.

The problem is an analysis of a tensile test (unit cube) using "test data" and the ANSYS Curve Fit to determine the constant in the Blatz–Ko material model. One eight-node brick is used with unit side lengths since one element is an exact model of the tensile test (Figure 15.31).

The experimental data for a tensile test must be entered manually or placed in a text file. The first column must be the extension ($\alpha_1 - 1$) and the second column, separated by a space or comma, must be the nominal stress (P_{11}). In the following example, the file BK.txt has a content that was calculated using the Blata–Ko model with $\mu = 100$ as follows:

0 0
0.1 20.22
0.2 33.42
0.3 42.19

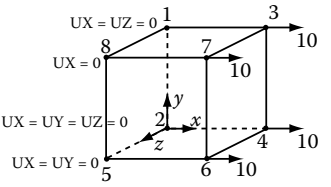


FIGURE 15.31 3D tensile test: fixed load.

The axial load is ramped up to $P_{11} = 40$. Minimum supports are imposed to prevent rigid displacement.

a. Activate ANSYS and set analysis type to structural.
See example 15.3.

b. Establish element type, material properties, and mesh the object.

```
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
HYPERELASTIC
3D 8 NODE 185              [this is SOLID185 element]
OK
CLOSE
PREPROCESSOR > MATERIAL PROP > MATERIAL MODEL
STRUCTURAL
NONLINEAR
ELASTIC
HYPERELASTIC
CURVE FITTING              [define material behavior window]
READ FROM FILE             [directory listing]
C BK.txt                   [file containing tensile test data]
OPEN                       [data are read in]
NEXT
CURVE FITS
HYPERELASTIC
BLATZ–KO (FOAM)
SOLVE                      [returns μ = 100.01]
OK                         [curve fit done]
PLOT                       [stress–strain curve]
SAVE AND CLOSE             [material constant for the model]
MATERIAL > EXIT
PREPROCESSOR > MODELING > CREATE > VOLUMES
BLOCK > BY DIMENSIONS
T 0 1, 0 1, 0 1 for X1, X2, Y1, Y2, Z1, Z2
C OK and the region appears showing the x–y plane
PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
OK
PREPROCESSOR > MESHING > MESH TOOL
C SIZE CONTROLS: GLOBAL > SET
C box for NDIV
T 1 in NDIV box            [one element is exact for the uniform
                           extension]
OK
C SHAPE: HEX               [this forces a brick instead of the default
                           tetrahedron]
MESH
```

PICK ALL (node numbers appear)
CLOSE (Mesh Tool)

c. Apply boundary conditions.

First, rotate the element so that you can see the base ($x = 0$) with nodes 1-2-5-8. This can be done in the dynamic model mode, which is interactive: on the right-hand side is a menu with 3D pictures of a block. Click on the button at the end of the list that shows two elements (Dynamic Model Mode). The cursor changes to a rotation symbol. Hold down the right mouse button to rotate the element interactively until you can see the face 1-2-5-8.

LOADS > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON NODES
C on node 2 (at the origin)
APPLY
ALL DOF
C on box for VALUES
T 0 for the value of the displacement components
APPLY
REPEAT THIS PROCESS TO SET UX AND UZ TO ZERO AT NODE 1
REPEAT THIS PROCESS TO SET UX AND UY TO ZERO AT NODE 5
REPEAT THIS PROCESS TO SET UX TO ZERO AT NODE 8
[This prevents rigid body motion and keeps the base in the Y–Z plane, but allows contraction.]
ROTATE THE ELEMENT so that you can see the face $x = 1$ with nodes 3-4-6-7
DEFINE LOADS > APPLY > STRUCTURAL > FORCE > ON NODES
C ON EACH NODE 3-4-6-7
OK
T 10 for the value of FX [$P_{11} = 40$]
OK

d. Solve for displacements and stresses.

The total load is specified, then reached by ramping up in small time steps. Results are saved after each time step.

SOLUTION > UNABRIDGED MENU
ANALYSIS TYPE > ANALYSIS OPTIONS
Set NLGEOM ON [this is for $\mathbf{K}_0$]
Set Stress Stiff ON (SSTIF at the bottom) [this is for $\mathbf{K}_1$]
OK
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE
File write frequency: select "every substep"
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME/TIME STEP
Time at end of load step: enter 1
Time step size: enter 0.1
Select RAMPED

Automatic time stepping: ON
Minimum time step size: enter 0.05
Maximum time step size: enter 0.1
OK
SOLUTION > SOLV CURRENT LS
CLOSE information window
C OK in solve window
(Convergence information is displayed as solution is marched out.)
CLOSE information window when solution is done

e. Display results: List displacements of selected nodes after each time step.

GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
DEF & UNDEF
OK and see deformed shape
GENERAL POST PROC > LIST RESULTS
NODAL SOLUTION
DOF SOLUTION > X-Component of displacement
OK [you should have UX = 0.271 at node 4]
CLOSE
ELEMENT SOLUTION
STRESS > X-COMPONENT OF STRESS
OK [you should have SX = 45.1; this is the true stress T_{11}]
CLOSE
TIME HIST POSTPROC
Close window
DEFINE VARIABLES
ADD
Select Nodal DOF Result
OK
T 4 in the box for "list of items" [this is another way of picking the node]
OK
Enter name: UX
Select Translation UX
OK
TIME HIST POSTPROC > DEFINE VARIABLES
ADD
ELEMENT RESULTS
OK
T 1 for the list of items (this is the element number)
OK
T 4 for the list of items (this is the node number for stress output)
OK
Enter name SX

Select STRESS > X-DIRECTION SX
OK
CLOSE (variable 2 is UX and variable 3 is SX)
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 3
For 2nd variable to list: enter 2
OK (Lists SX and UX for each time step)
CLOSE

f. Plot results: plot the displacement at each time.
TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable No.: enter 2 [UX on *x*-axis]
OK
TIME HIST POSTPROC > GRAPH VARIABLES
For 1st variable, enter 3 [SX on *y*-axis]
OK
Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: UX
Enter y-axis label: SX
SELECT: SPECIFIED X-RANGE
ENTER X-RANGE OF 0 TO 0.3
OK
Utility Menu: PLOT > REPLOT
This graph can be printed for a report after adjusting colors

15.38 CURVE FIT FOR NONLINEAR ELASTICITY USING POLYNOMIAL MODEL

This is an analysis of a tensile test (unit cube) using test data and the ANSYS Curve Fit to determine the constants in the first-order polynomial model. One 8-node brick is used with unit side lengths since one element is an exact model of the tensile test (Figure 15.32).

The experimental data for a tensile test must be entered manually or placed in a text file. The first column must be the extension ($\alpha_1 - 1$) and the second column, separated by a space or comma, must be the nominal stress (P_{11}). In the following example, the file Uni.txt has content

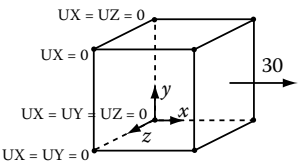


FIGURE 15.32 3D tensile test: fixed stress.

0 0
0.0341 9.71
0.0697 18.86
0.1068 27.46
0.1453 35.55
0.1850 43.13

These data are taken from Table 13.1.

Experimental data for volumetric strain must list the volume ratio (J) in the first column and the mean pressure (p) in the second column. If no volumetric strain data are entered, the model is assumed to be incompressible. The file Vol.txt has the content

0.98 10
0.96 20
0.94 30

which is the exact values for $\kappa = 500$ or $d = 0.004$.

The axial stress is ramped up to $T_{11} = 30$. Minimum supports are imposed to prevent rigid displacement.

a. Activate ANSYS and set analysis type to structural.
See example 15.3.

b. Establish element type.
PREPROCESSOR > ELEMENT TYPE > ADD
ADD
HYPERELASTIC
C 3D 8 NODE 185 [this is SOLID185 element]
OK
CLOSE

c. Material properties by a curve fit.
PREPROCESSOR > MATERIAL PROP > MATERIAL MODEL
STRUCTURAL
NONLINEAR
ELASTIC
HYPERELASTIC
CURVE FITTING [define material behavior window]
READ FROM FILE [directory listing]
C Uni.txt [file containing tensile test data]
OPEN [data are read in]
NEXT [table for biaxial data appears]
NEXT
OK [no biaxial data warning]
NEXT [table for shear data appears]
OK [no shear data]
NEXT [table for simple shear data appears]

OK [no data for simple shear]
NEXT [table for volumetric test data appears]
READ FROM FILE
C Vol.TXT [file containing volumetric data]
OPEN
NEXT
HYPERELASTIC
POLYNOMIAL
C 1st ORDER
SOLVE [returns $C_{10} = 21.9$, $C_{01} = 28.1$, $d = 0.004$]
OK [curve fit done]
PLOT [stress–strain curves]
SAVE AND CLOSE [material constants for the model]
MATERIAL > EXIT

d. Create geometry.

PREPROCESSOR > MODELING > CREATE > VOLUMES > BLOCK > BY DIMENSIONS
T 0 1, 0 1, 0 1 for X1,X2,Y1,Y2, Z1,Z2
C OK and the region appears showing the x–y plane
PLOT CTRLS > NUMBERING
C box after Node numbers to turn them ON
OK

e. Mesh as one element.

PREPROCESSOR > MESHING > MESH TOOL
SIZE CONTROLS > GLOBAL > SET
C box for NDIV
T 1 in NDIV box [one element is exact for the uniform extension]
OK
SHAPE: HEX [this forces a brick instead of the default tetrahedron]
MESH
PICK ALL (node numbers appear)
CLOSE (Mesh Tool)

f. Apply boundary conditions.

First, rotate the element so that you can see the base ($x = 0$) with nodes 1-2-5-8. This can be done in the dynamic model mode, which is interactive: on the right-hand side is a menu with 3D pictures of a block. Click on the button at the end of the list that shows two elements (Dynamic Model Mode). The cursor changes to a rotation symbol. Hold down the right mouse button to rotate the element interactively until you can see the face 1-2-5-8.

LOADS > DEFINE LOADS > APPLY > STRUCTURAL
DISPLACEMENT > ON NODES
C on node 2 (at the origin)
APPLY
C ALL DOF
C on box for VALUES

T 0 for the value of the displacement components
C APPLY
REPEAT THIS PROCESS TO SET UX AND UZ TO ZERO AT NODE 1
REPEAT THIS PROCESS TO SET UX AND UY TO ZERO AT NODE 5
REPEAT THIS PROCESS TO SET UX TO ZERO AT NODE 8
[This prevents rigid body motion and keeps the base in the Y–Z plane, but allows contraction.]
ROTATE THE ELEMENT so that you can see the face $x = 1$ with nodes 3-4-6-7
DEFINE LOADS > APPLY > STRUCTURAL > PRESSURE > ON AREAS
C ON THE FACE 3-4-6-7
OK
T –30 for the value [$T_{11} = 30$, negative pressure]
C OK [close warning window]

g. Solve for displacements and stresses.
The total load is specified, then reached by ramping up in small time steps. Results are saved after each time step.
SOLUTION > UNABRIDGED MENU
ANALYSIS TYPE > ANALYSIS OPTIONS
Set NLGEOM ON [this is for $\mathbf{K}_0$]
Set Stress Stiff ON (SSTIF at the bottom) [this is for $\mathbf{K}_1$]
OK
SOLUTION > LOAD STEP OPTS > OUTPUT CTRLS > DB/RESULTS FILE
File write frequency: select "every substep"
OK
SOLUTION > LOAD STEP OPTS > TIME/FREQUENC > TIME/TIME STEP
Time at end of load step: enter 1
Time step size: enter 0.1
Select RAMPED
Automatic time stepping: ON
Minimum time step size: enter 0.05
Maximum time step size: enter 0.1
OK
SOLUTION > SOLV CURRENT LS
CLOSE information window
C OK in solve window
(Convergence information is displayed as solution is marched out.)
CLOSE information window when solution is done

h. Display results: List displacements of selected nodes after each time step.
GENERAL POST PROC > PLOT RESULTS > DEFORMED SHAPE
DEF & UNDEF
OK and see deformed shape
GENERAL POST PROC > LIST RESULTS

NODAL SOLUTION
DOF SOLUTION > X-Component of displacement
OK [you should have UX = 0.1149 at node 4]
CLOSE
ELEMENT SOLUTION
STRESS > X-COMPONENT OF STRESS
OK [you should have SX = 30.00; this is the true stress T_{11}]
CLOSE
TIME HIST POSTPROC
Close window
DEFINE VARIABLES
ADD
Select Nodal DOF Result
OK
T 4 in the box for "list of items" [this is another way of picking the node]
OK
Enter name: UX
Select Translation UX
OK
TIME HIST POSTPROC > DEFINE VARIABLES
ADD
ELEMENT RESULTS
OK
T 1 for the list of items (this is the element number)
OK
T 4 for the list of items (this is the node number for stress output)
OK
Enter name SX
Select STRESS > X-DIRECTION SX
OK
CLOSE (variable 2 is UX and variable 3 is SX)
TIME HIST POSTPROC > LIST VARIABLES
For 1st variable to list: enter 3
For 2nd variable to list: enter 2
OK (Lists SX and UX for each time step)
CLOSE

i. Plot results: plot the displacement at each time.

TIME HIST POSTPROC > SETTINGS > GRAPH
x-Axis variable: select Single Variable
Single variable no.: enter 2 [UX on *x*-axis]
OK
TIME HIST POSTPROC > GRAPH VARIABLES

For 1st variable, enter 3 [SX on *y*-axis]
OK
Utility Menu: PLOT CTRLS > STYLE > GRAPHS > MODIFY AXES
Enter x-axis label: UX
Enter y-axis label: SX
SELECT: SPECIFIED Y-RANGE
ENTER Y-RANGE OF 0 TO 50
SELECT: SPECIFIED X-RANGE
ENTER X-RANGE OF 0 TO 0.2
OK
Utility Menu: PLOT > REPLOT
This graph can be printed for a report after adjusting colors

BIBLIOGRAPHY

Alawadhi, E. M., *Finite Element Simulations Using ANSYS*, CRC Press, Boca Raton, FL, 2010.

Madenci, E., and I. Guven, *The Finite Element Method and Applications in Engineering Using ANSYS*, Springer, New York, 2006.

Moaveni, S., *Finite Element Analysis, Theory and Applications with ANSYS*, 3rd ed., Prentice Hall, Upper Saddle River, NJ, 2008.

Stolarski, T. A., Y. Nakasone, and S. Yoshimoto, *Engineering Analysis with ANSYS Software*, Elsevier, New York, 2006.

16 ANSYS Workbench

This book is mainly about understanding the foundations of finite element analysis. But the ultimate goal is, of course, to apply the finite element method (FEM) to real material bodies. This means application to complex geometries. Workbench provides the means.

Workbench is a fully developed Computer Aided Engineering package. The examples in Chapter 15 use the original ANSYS program that has been renamed ANSYS APDL. The tools for constructing the geometry of the material body using APDL are somewhat rudimentary. For complicated figures, you can use a CAD program to draw the figure and then import the geometry to ANSYS. Workbench eliminates that step. It has all of the tools of a CAD program and will automatically interface with the finite element analysis.

Workbench is especially useful for analyses that involve interaction of solid mechanics, heat transfer, fluid mechanics, and electrodynamics of materials, so called multiphysics problems. However, we will only consider here solid mechanics and heat conduction.

Product demonstrations, animated tutorials, and training materials are provided by ANSYS for existing customers through the Customer Portal: http://www1.ansys.com/customer/. This is a good place to start, but the instructions are not very detailed. New users should utilize one of the sources listed in the appended bibliography.

16.1 TWO- AND THREE-DIMENSIONAL GEOMETRY

The basic mouse operations are as follows. The left mouse button is used to click a single selection; click and hold to sweep for a continuous selection, or combine with the control key for multiple selection. The middle button is used for a rotation by click and hold, or a scroll wheel can be used to zoom in or out. The right mouse button is clicked to obtain a context menu, or used to click and drag for a box zoom.

The program is initiated by

Start > All Programs > Ansys 12.1 > Workbench

After awhile the Workbench window appears with menu bars along the top, a graphics window (project schematic window) on the right, and a toolbox menu on the left.

Click on the + Sign to Expand the Component System Menu
Double Click Component Systems > Geometry

This introduces into the project schematic window a schedule of tasks. In this case, the only task is to create a geometric figure.

Double Click on the Geometry task (not the Geometry heading)

This starts the software package called the Design Modeler within which the 2D or 3D body can be constructed. A menu appears in which one must select the units to be used for the drawing. Select the units and click OK. Then click Sketching in the lower left menu box to get the Sketching toolbox menu with submenus for Draw, Modify, Dimensions, Constraints, and Settings. Each menu has icons and names that are more or less self-explanatory. Their use will be demonstrated in the following examples. The most commonly used Draw Tools are the Line Tool for straight lines, and the Polyline Tool for open or closed polygonal figures. Click on a point for each apex, but do not drag out the line by holding down the mouse button. A rough figure is sketched with the Draw Tools without concern for the dimensions.

The Constraints Toolbox contains tools for restricting Symmetry of figures, identifying Parallel Lines, lines of Equal Length, and so forth. Certain constraints are automatic: An H will appear if the line is horizontal; a V will appear if a line is vertical; a C will appear if the selected point falls on an existing line; a P will appear if a selected point is coincident with an existing point; etc.

The Dimensions Toolbox is used to impose the desired dimensions on the figure. The figure is drawn approximately. Next, one typically picks a line or a point for the origin of a dimension and then another point for the end of the dimension, then a point for the location of the dimension line. The value of the dimension is entered in the Detail Box, and the size of the figure is automatically adjusted. The Modify Toolbox can be used to insert fillets at corners, trim away extra construction lines, extend the drawing by Replicate of parts, and so forth.

A three-dimensional drawing is typically constructed by Extrude, Revolve, or Sweep of a 2D sketch. For plane stress, plane strain, plate, or shell analysis, it is necessary to first associate a surface with the 2D sketch of the body.

16.2 STRESS ANALYSIS

In the startup toolbox, the Analysis Systems menu contains a list of tasks that can be executed using Workbench. If one double clicks on Static Structural (ANSYS), for example, a menu of subtasks to be performed appears in the Project Schematic (Graphics) window:

Static Structural (ANSYS)
Engineering Data
Geometry
Model
Setup
Solution
Results

The Engineering Data task includes the selection of a material.

If a geometric figure has been previously constructed, this task is opened first and then associated with the stress analysis by dragging the geometry to the Geometry task. A double click on Model starts the Mechanical task in which the finite element analysis is formulated. The desired output is set up in the Solution task.

The choice of element type and the mesh layout can be automatically generated by using default parameters of Workbench. The element type used and other solution information is available by clicking the

Solution > Solution Information box

after completing the solution phase.

The element types used are as follows:

Plane Stress, Plane Strain, Axisymmetric: Plane 182, Plane 183.
3D Solid Bodies: Solid95, Solid186, Solid92 (tetrahedral), Solid187 (tetrahedral).
Plates and Shells: Shell181.
Beam and Column: Beam188.

The default mesh can be changed in a number of ways with available menus. Workbench will also do the work of improving the mesh for you by automatically refining the mesh in regions of high strain and repeating the analysis until a convergence criterion is satisfied.

16.3 SHORT BEAM EXAMPLE

A thin rectangular sheet is to be analyzed as a plane stress problem (Figure 16.1): $a = 100$ mm, $b = 100$ mm, $t = 1$ mm.

16.3.1 Short Beam Geometry

In preparation for the analysis, we will sketch the region and create a 2D model.

START > ALL PROGRAMS > ANSYS 12.1 > WORKBENCH	[wait for window]
C COMPONENT SYSTEMS	[on the + sign to expand the menu]

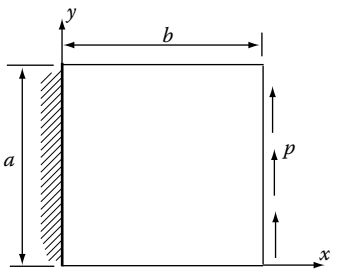


FIGURE 16.1 Short beam.

CC GEOMETRY	[start Project A]
CC GEOMETRY	[on the ? mark, to start Design Modeler]
Select UNITS = MILLIMETER	
C OK	
C SKETCHING	[left side below tree outline]
C LOOK AT FACE PLANE	[right most logo on menu bar]
C DRAW to get menu	[in sketching toolbox, if needed]
C RECTANGLE	
Click on the origin and the approximate location of the upper right corner	
C DIMENSIONS	[to get menu, General is selected]
C top edge and location for dimension	
C left edge and location for dimension	
Enter dimensions in the Details View	[100 for each then ENTER]
C CONCEPT > SURFACES FROM SKETCH	[top menu bar]
EXPAND XYPLANE	[click on the + next to x–y plane in the tree]
C SKETCH1	
C APPLY for base object	[first click yellow region if necessary]
Enter THICKNESS = 1	
C GENERATE	[body is then shaded]
C ZOOM TO FIT	[logo bar]
C FILE > CLOSE DESIGN MODELER	[project schematic is visible]
C FILE > SAVE	[choose a file name, e.g., sheet]
T sheet	
C SAVE	[project saved as sheet.wbpj]
C FILE > EXIT	

16.3.2 Short Beam, Static Loading

A thin rectangular sheet is to be analyzed as a linear elastic plane stress problem (Figure 16.1): $a = 100$ mm, $b = 100$ mm, $t = 1$ mm, $p = 1$ MPa. The material is structural steel: $E = 2 \times 10^5$ MPa, $\nu = 0.3$. These are the default values in Workbench. The geometry has been stored in a file with name "sheet."

a. Start Workbench and retrieve stored geometry file.

START > ALL PROGRAMS > ANSYS 12.1 > WORKBENCH	[wait for window]
FILE > OPEN	

C sheet	[or the file name from 16.3.1]
C OPEN	[the geometry appears as project A]
RC GEOMETRY	
C PROPERTIES	[if a short menu appears close properties and repeat this step]
Set ANALYSIS TYPE = 2D	[on pull down menu]
CLOSE PROPERTIES	[X in upper right corner]
WINDOW	
CC STATIC STRUCTURAL	[appears as project B]
C GEOMETRY(A) and DRAG onto GEOMETRY(B)	[a link appears]

b. Apply loads and supports for 2D analysis.

CC MODEL	[mechanical starts]
C UNITS	[choose mm,kg,N]
C STATIC STRUCTURAL	[project tree]
C SUPPORTS > FIXED SUPPORT	[menu bar for Supports]
C EDGE SELECTION LOGO	[logo bar]
C LEFT EDGE	
C APPLY	[details menu, tag appears]
C LOADS > FORCE	[menu bar for Loads]
C RIGHT EDGE	[for uniformly distributed load]
C APPLY	[details menu, tag appears]
C DEFINE BY	
C COMPONENTS	[vector drop menu]
C Y-COMPONENT	[drop menu]
ENTER 100®	[F = pat = $10^6 \times 10^{-1} \times 10^{-3}$ N = 100 N]
C STATIC STRUCTURAL in Tree to see loads and supports	

c. Specify the desired output and solve using the default mesh.

C SOLUTION	[in tree outline]
C DEFORMATION > DIRECTIONAL	[menu bar for Displacement]
C ORIENTATION	[details menu]
Select Y AXIS on details menu	[drop down menu]
C SOLVE	[contour plot appears of UY]
C PROBE	[menu bar]
[zoom in as necessary]	
C upper right corner	[Note maximum UY = 0.00369 mm]
C EDGES > SHOW ELEMENTS	[logo bar, to see FE grid]
C SOLUTION INFORMATION in tree	[to see info on element type and nodes]
C FILE > CLOSE MECHANICAL	
C FILE > EXIT	

16.3.3 Short Beam, Transient Analysis

A thin rectangular sheet is to be analyzed as a linear elastic plane stress problem (Figure 16.1): $a = 100$ mm, $b = 100$ mm, $t = 1$ mm, $p = 1$ MPa. The material is structural steel: $E = 2 \times 10^5$ MPa, $\nu = 0.3$, $\rho = 7850$ kg/m^3. The load is applied as a step load. The geometry has been stored in a file with name "sheet."

a. Start Workbench and retrieve geometry.	
START > ALL PROGRAMS >	[wait for window]
ANSYS 12.1 > WORKBENCH	
C FILE > OPEN	
C sheet	[file with saved geometry]
C OPEN	
b. Start transient structural analysis project.	
CC TRANSIENT STRUCTURAL	[analysis systems toolbox]
Drag Geometry A to Geometry box of	
Transient Structural schematic B	[a link line appears]
c. Specify 2D analysis.	
RC GEOMETRY	[for new project]
C PROPERTIES	
Set ANALYSIS TYPE = 2D	[on pull down menu]
CLOSE PROPERTIES WINDOW	[X in upper right corner]
d. Specify material properties.	
CC ENGINEERING DATA	
C STRUCTURAL STEEL	[if properties not displayed]
C VIEW > PROPERTIES if not displayed	[top menu bar]
	[Note that the density, modulus, and Poisson ratio have the desired values]
C RETURN TO PROJECT	[menu bar]
e. Add support and load conditions.	
CC MODEL	[mechanical starts]
C UNITS	[choose mm,kg,N]
C TRANSIENT	[project tree]
C SUPPORTS > FIXED SUPPORT	[menu bar]
C EDGE SELECTION LOGO	[menu bar]
C LEFT EDGE	
C APPLY	[details menu, tag appears]
C LOADS > FORCE	[menu bar]
C RIGHT EDGE	
C APPLY	[details menu, tag appears]
C DEFINE BY	
C COMPONENTS	[vector drop menu]
C Y COMPONENT	
ENTER 100®	[F = pat = $10^6 \times 10^{-1} \times 10^{-3}$ N]

f. Time and time steps.

C ANALYSIS SETTINGS [outline tree]
Enter STEP END TIME = 0.0002 [half period from frequency analysis]

C DEFINE BY
C SUBSTEPS [on drop menu]
Enter INITIAL SUBSTEPS = 100®
Enter MINIMUM SUBSTEPS = 100®
Enter MAXIMUM SUBSTEPS = 100®

g. Specify output to save and solve equations.

C SOLUTION [in tree outline]
C DEFORMATION > DIRECTIONAL [menu bar for Displacement]

C ORIENTATION
Select Y AXIS on drop menu
C SOLVE [contour map of UY displayed]
C GRAPH in comments window to see time history of max UY.
Note maximum UY = 7.0082×10^{-3} at $t = 1.04 \times 10^{-4}$.
C ANIMATION ARROW to see motion
C ANIMATION STOP
Click SOLUTION INFORMATION for data on elements and damping
C FILE > CLOSE MECHANICAL
C FILE > EXIT

16.4 FILLETED BAR EXAMPLE

The filleted bar shown in Figure 16.2 is loaded in tension. The material is structural steel. The analysis is plane stress. The maximum stress is to be determined.

a. Sketch the region.

START > ALL PROGRAMS > ANSYS 12.1 > WORKBENCH [wait for window]
CC STATIC STRUCTURAL
RC GEOMETRY
C NEW GEOMETRY
Select UNITS = MILLIMETER

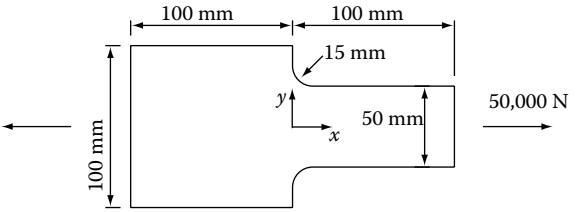


FIGURE 16.2 Filleted bar.

C OK
C SKETCHING [left side below tree outline]
C LOOK AT FACE PLANE [right most logo on menu bar]
C DRAW to get menu [sketching toolbox]
C POLYLINE
Click on approximate location of each of the eight corners of the object (before fillets) in sequence. Do not click on the starting point again. Be sure that an H appears for horizontal lines and a V for vertical lines.
RC after the last corner to get a menu and then select
C CLOSED END
C CONSTRAINTS [sketching toolbox]
C EQUAL LENGTH
C PAIRS OF LINES that are required to have equal length [repeat these two steps for each pair, two pairs of horizontal lines and one pair of vertical lines]
C SYMMETRY [constraint menu]
C X-AXIS [axis of symmetry]
C a top and bottom pair of lines
C DIMENSIONS to get menu [general is default]
C on a line and the location for the dimension [for each dimension]
ENTER dimensions in details menu
C ZOOM TO FIT LOGO [if necessary]
C DISPLAY [dimensions menu]
C to deselect NAME and VALUE is selected automatically [to display actual values]
C MODIFY [sketching toolbox]
C FILLET
Enter Radius = 15®
C pairs of lines to create a fillets [C edge filter if necessary]
C DIMENSIONS > RADIUS
C on a fillet and drag dimension line normal to the fillet [shows $R = 15$]

b. Create a surface body.

C CONCEPT > SURFACES [detail window appears]
FROM SKETCH
Enter THICKNESS = 10®
EXPAND XY PLANE [click on the + next to x–y plane in the tree]
C SKETCH1
C BASE OBJECT NOT SELECTED [Apply option appears]
C APPLY [Base Object = 1 sketch]
C GENERATE [lightning logo—body is then shaded]

C FILE > CLOSE DESIGN MODELER [project schematic is visible]
RC GEOMETRY [wait]
C PROPERTIES
Set ANALYSIS TYPE = 2D [on details menu]
CLOSE PROPERTIES WINDOW

c. Apply loads and supports.

CC MODEL [wait for Mechanical to start]
C UNITS
C mm,kg,N
RC STATIC STRUCTURAL
C INSERT
C FRICTIONLESS SUPPORT
C EDGE SELECTION LOGO
C LEFT EDGE
C APPLY [details menu, tag appears]
RC STATIC STRUCTURAL
C INSERT
C FORCE [for uniformly distributed load]
C DEFINE BY [details menu]
C COMPONENTS [vector drop menu]
C X-COMPONENT to select it
Enter MAGNITUDE = 50000®
C RIGHT EDGE
C Yellow area by GEOMETRY in details window if necessary
C APPLY [tag appears]
C STATIC STRUCTURAL in Tree to see loads and supports [Rigid motion constraints will be automatically added]

d. Specify the desired output.

RC SOLUTION [in tree outline]
C INSERT
C STRESS > MAXIMUM PRINCIPAL

e. Mesh the body.

C MESH in the tree outline to highlight it
C MESH CONTROL > SIZING
C on body
C APPLY on geometry detail
C ELEMENT SIZE
Enter 10® [in place of "default"]
C MESH CONTROL > METHOD [menu bar]
C ANYWHERE ON THE BODY
C APPLY on geometry detail [geometry = 1 body]
C METHOD [details menu]

Select TRIANGLES from drop menu [tree shows "All Triangles Method"]
RC ALL TRIANGLES METHOD [in tree]
C GENERATE MESH
C MESH [too see mesh]
EXPAND SATISTICS to see the number of elements

f. Solve and view results.
C SOLVE [lightning logo on menu bar]
[Note warning that rigid motion has been prevented by weak springs]
C MAXIMUM PRINCIPAL STRESS [in solve tree to see contour plot]
[Note maximum is σ_1 = 147 MPa on fillet]
[Use BOX ZOOM to expand the region of high stress and Probe if desired]
FILE > CLOSE MECHANICAL
FILE > EXIT

16.5 SHEET WITH A HOLE

A sheet with a central hole (Figure 16.3) is stretched by a uniform edge stress S resulting in a stress concentration at the hole. Symmetry is used so that only the upper-right quadrant is retained. This demonstrates automatic mesh refinement to improve result. In any convenient system of units: S = 100, a = 20, b = 10, r = 5, structural steel. The stress concentration is sought.

a. Sketch the region.
START > ALL PROGRAMS > ANSYS 12.1 > WORKBENCH [wait for window]
CC STATIC STRUCTURAL
CC GEOMETRY [to start Design Modeler]
Select UNITS = MILLIMETER [or your choice]
C OK

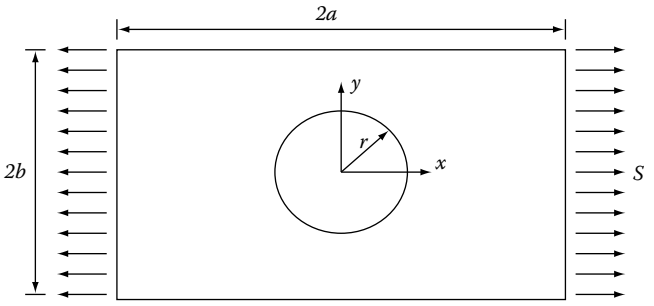


FIGURE 16.3 Sheet with a hole.

C SKETCHING [left side below tree outline]
C LOOK AT FACE PLANE [right most logo on menu bar]
C DRAW to get menu [sketching toolbox]
C RECTANGLE
Click on the origin and the approximate location of the upper right corner
C CIRCLE
C the origin and the approximate location of a point on the circle
C DIMENSIONS [to get menu, General is selected]
C top edge and location for dimension
C right edge and location for dimension
C RADIUS
C on circle and location for dimension
Enter dimensions in the Details View
C MODIFY
C TRIM > IGNORE AXIS
Click on each line segment (circle and two enclosed line segments) to trim away [we now have the upper-right quadrant sketched]

b. Create a surface body.

C CONCEPT > SURFACES FROM SKETCH [detail window appears]
Enter THICKNESS = 1® [arbitrary choice (area = 10)]
EXPAND XYPLANE [click on the + next to x–y plane in the tree]
C SKETCH1
C APPLY [first click yellow region if necessary]
C GENERATE [body is then shaded]
C FILE > CLOSE DESIGN MODELER [project schematic is visible]
RC GEOMETRY [WAIT]
C PROPERTIES
Set ANALYSIS TYPE = 2D [on pull down menu]
CLOSE PROPERTIES WINDOW [X in upper right corner]

c. Apply loads and supports.

CC MODEL [Wait for the model to appear]]
C UNITS [choose mm,kg,N, or your choice]
C STATIC STRUCTURAL
C SUPPORTS > FRICTIONLESS SUPPORT [menu bar]
C EDGE SELECTION LOGO [menu bar]
C LEFT EDGE
C APPLY [tag appears]

C SUPPORTS > FRICTIONLESS SUPPORT [menu bar]
C BOTTOM EDGE
C APPLY [tag appears]
C LOADS > PRESSURE [menu bar]
C RIGHT EDGE
C APPLY [tag appears]
Enter MAGNITUDE = −1® [SX = +1]
C STATIC STRUCTURAL in Tree to see loads and supports

d. Specify the desired output and solve using the default mesh.

C SOLUTION [in tree outline]
C STRESS > NORMAL [*X*-direction is default]
C SOLVE
C NORMAL STRESS [in solution tree to see contour plot]
C EDGES > SHOW ELEMENTS [menu bar]
[note the relatively coarse grid in the region of high stress and the maximum value of the stress]

e. Automatically refine the mesh to improve the solution.

RC NORMAL STRESS
C INSERT > CONVERGENCE
C SOLUTION
SET MAX REFINEMENT LOOP TO 5
C CONVERGENCE
ENTER ALLOWABLE CHANGE = 1® [1% improvement in answer is sought]
C SOLVE [solution is repeated with a new mesh up to five times]
[mesh statistics and new max stress is displayed]
C NORMAL STRESS [to see refined elements and better value of maximum stress]

BIBLIOGRAPHY

All of these are published by SDC Publications, Mission, KS (Schroff Development Corporation, www.schroff.com)

Dadkhah, F., and J. Zecher, *ANSYS Workbench Software, Tutorial with Multimedia CD, Release 12*, SDC Pub., 2009.

Lawrence, K. L., *ANSYS Workbench Tutorial, Structural & Thermal Analysis Using the ANSYS Workbench Release 12.1 Environment*, SDC Pub., 2010.

Lee, H.-H., Finite Element Simulations with ANSYS Workbench 12, SDC Pub., 2010.

Mechanical Engineering

The Finite Element Method for Mechanics of Solids with ANSYS Applications

ELLIS H. DILL

While the finite element method (FEM) has become the standard technique used to solve static and dynamic problems associated with structures and machines, ANSYS software has developed into the engineer's software of choice to model and numerically solve those problems.

An invaluable tool to help engineers master and optimize analysis, *The Finite Element Method for Mechanics of Solids with ANSYS Applications* explains the foundations of FEM in detail, enabling engineers to use it properly to analyze stress and interpret the output of a finite element computer program such as ANSYS.

Illustrating presented theory with a wealth of practical examples, this book covers topics including

- Essential background on solid mechanics (including small- and large-deformation elasticity, plasticity, viscoelasticity) and mathematics
- Advanced finite element theory and associated fundamentals, with examples
- Use of ANSYS to derive solutions for problems that deal with vibration, wave propagation, fracture mechanics, plates and shells, and contact

Totally self-contained, this text presents step-by-step instructions on how to use ANSYS Parametric Design Language (APDL) and the ANSYS Workbench to solve problems involving static/dynamic structural analysis (both linear and nonlinear) and heat transfer, among other areas. It will quickly become a welcome addition to any engineering library, equally useful to students and experienced engineers.

K12048
ISBN: 978-1-4398-4583-7
90000
9 781439 845837
www.crcpress.com

CRC Press
Taylor & Francis Group
an informa business

6000 Broken Sound Parkway, NW
Suite 300, Boca Raton, FL 33487
711 Third Avenue
New York, NY 10017
2 Park Square, Milton Park
Abingdon, Oxon OX14 4RN, UK