Lecture Notes:

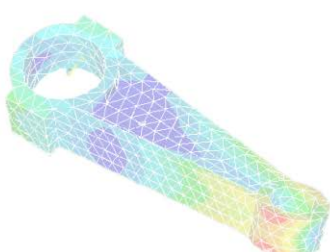
An Introduction to the Finite Element Method

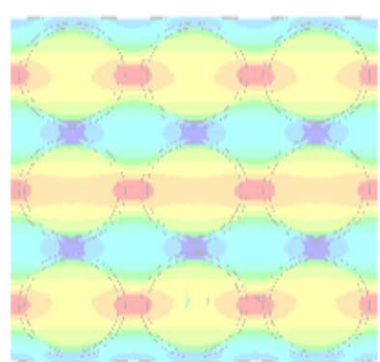
Yijun Liu

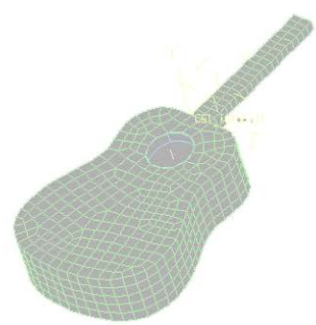
CAE Research Laboratory
Mechanical Engineering
University of Cincinnati
Cincinnati, OH 45221-0072, U.S.A.

E-mail: Yijun.Liu@uc.edu
Web: www.yijunliu.com

(*Last Updated*: October 21, 2015)







Copyright Notice

Table of Contents

Preface

This is a basic introduction to the finite element method (FEM) for undergraduate students in engineering or other readers who have no previous experience with the FEM. The lecture notes cover the basic concepts in the FEM using the simplest mechanics problems as examples, and lead to the discussions and applications of the one-dimensional (1-D) bar and beam, 2-D plane stress and plane strain, plate and shell, and 3-D solid elements in the analyses of structural stresses, vibrations and dynamics. The understanding of the FEM principles and procedures, and correct usage of the FEM software are emphasized throughout the notes.

These lecture notes have been developed by the author for the undergraduate courses on the FEM in the Mechanical Engineering Department at the University of Cincinnati since 1997. The materials in these notes are aimed for students in mechanical, civil and aerospace engineering, who need a general background in the FEM so that they can apply the FEM in their design and analysis of components, structures or systems using available commercial FEM software. For students who will conduct research on the FEM, these lecture notes should only serve as an introduction and they should consult to the references listed at the end of the notes for more rigorous treatment of the subject in order to have the necessary theoretical background and programming skills in developing new capabilities in the FEM.

The lecture notes include eight chapters and can be used for an undergraduate FEM course in one semester (15 weeks with three one-hour lectures each week) or in two quarters (20 weeks with three one-hour lectures each week, with four or five accompanying computer laboratory sessions each quarter). Chapter 1 gives a basic introduction to the concept of the FEM using the spring system as examples. It also reviews the matrix algebra that is essential for the FEM. Chapter 2 introduces the bar and beam elements and outlines the general procedures in the formulations and application of the FEM. Chapter 3 covers 2-D problems in elasticity, that is, plane stress and plane strain elements. Chapter 4 discusses various modeling techniques in the FEM and related topics, such as error indicators and how to evaluate the FEM results. Chapter 5 gives an introduction to the plate and shell elements, emphasizing the correct use of these types of elements. Chapter 6 provides the formulations and applications of the FEM in general 3-D elasticity problems. Chapter 7 is an introduction of the FEM in structural vibration and dynamics analysis, covering normal modes, harmonic and transient responses of structures using the FEM. Chapter 8 covers the basics in thermal analysis of structures using the FEM. Exercise problems and/or projects using FEM software packages are provided at the end of each chapter. Further readings are provided in the Reference section to conclude the lecture notes.

The author thanks many of his former undergraduate and graduate students at the University of Cincinnati for their suggestions on the earlier versions of these lecture notes and for their contributions to many of the examples used in the lecture notes.

Yijun Liu

Cincinnati, Ohio, USA
Winter 2007

Chapter 1. Introduction

I. Some Basic Concepts

A Simple Idea

The *finite element method* (FEM), or *finite element analysis* (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life (see, e.g., Figure 1.1), as well as in engineering. For example, children play the toy Lego by using many small pieces of simple geometries to build various objects such as trains, ships or buildings. With more and smaller pieces, these objects often look more realistic. As another example, a digital image, which looks smooth and colorful, is in fact composed of millions of dots that just have one simple color.

(a)

(b)

Figure 1.1. Objects built with simple and small pieces: (a) a fire engine built with Lego®; and (b) a house built with many elements – bricks, beams, columns, panels and so on.

In mathematical terms, this is simply the use of the limit concept, that is, to approach or represent a smooth objects with a finite number of simple pieces and increasing the number of such pieces will increase the accuracy of this representation. For example, ancient people used this concept to estimate the area of a circle well before the formula $A = \pi R^2$ was established (where R is the radius of the circle). In doing so, a circle is approximated by a polygon or divided into many triangles (Figure 1.2). The area of one triangle is given by:

$$S_i = \frac{1}{2} R_i L_i ,$$

where R_i is the height and L_i the base length of the triangle. The area A of the circle can be therefore obtained in the following manner:

$$S_N = \sum_{i=1}^{N} S_i = N\left(\frac{1}{2} R_i L_i\right) \rightarrow \frac{1}{2} R L_{total} = \pi R^2 = A, \; as \; N \rightarrow \infty ,$$

where N is the total number of triangles (or *elements*) and $L_{total} = N L_i$ is, in the limit, the circumference of the circle, which is $2\pi R$ as it is known now.

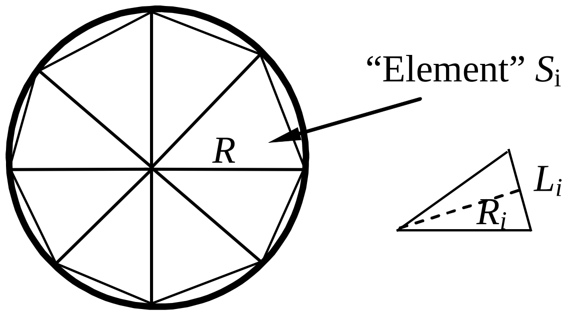


Figure 1.2. Approximation of the area of a circle using a finite number of triangles.

From the above mentioned examples, one can conclude that objects with complicated geometries can be represented by many small pieces (or *elements*) with simple geometries. As the number of such pieces increase, the representation becomes more accurate. This is exactly the same concept used in the development of the FEM as one will see in later chapters.

Why the Finite Element Method?

Computers have revolutionized the practice of engineering. Design of a product that used to be done by tedious hand drawings has been replaced by computer-aided design (CAD) using computer graphics. Analysis of a design used to be done by hand calculations and many of the testing have been replaced by computer simulations using computer-aided engineering (CAE) software. Together, CAD, CAE and CAM (computer-aided manufacturing) have dramatically changed the landscape of engineering. For example, a car, that used to take five to six years from design to product, can now be produced starting from the concept design to the manufacturing within about 18 months using the CAD/CAE/CAM technologies. A company without adopting the CAD/CAE technologies is deemed to lose ground in the competitive market place. FEM is the most widely applied simulation tool in CAE or one of the most powerful calculators available for engineering students.

Applications of the FEM in Engineering

There are numerous applications of the FEM in industries today and below is only a very short list:

- Mechanical/Aerospace/Automobile/Civil/Electrical Engineering
- Structure stress and dynamic analysis
- Thermal/fluid flows
- Electrostatics/Electromagnetics
- Geomechanics
- Biomechanics

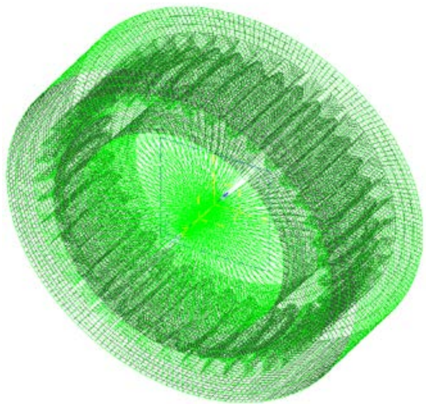


Figure 1.3. Modeling of gear coupling using the FEM.

A Brief History of the FEM

An account of the historical development of finite element method and the computational mechanics in general was given by O. C. Zienkiewicz recently, which can be found in Ref. [1]. A few major milestones are as follows:

1943 ----- Courant (Variational methods which laid the foundation for FEM)

1956 ----- Turner, Clough, Martin and Topp (Stiffness method)

1960 ----- Clough (Coined "Finite Element", solved plane problems)

1970s ----- Applications on "mainframe" computers

1980s ----- Microcomputers, development of pre- and postprocessors (GUI)

1990s ----- Analysis of large structural systems, nonlinear and dynamic problems

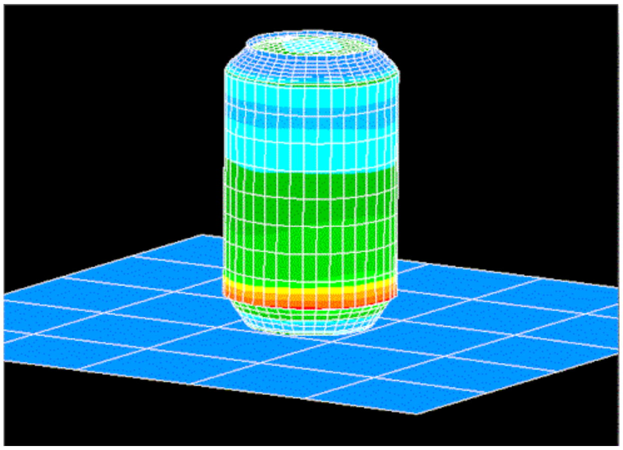
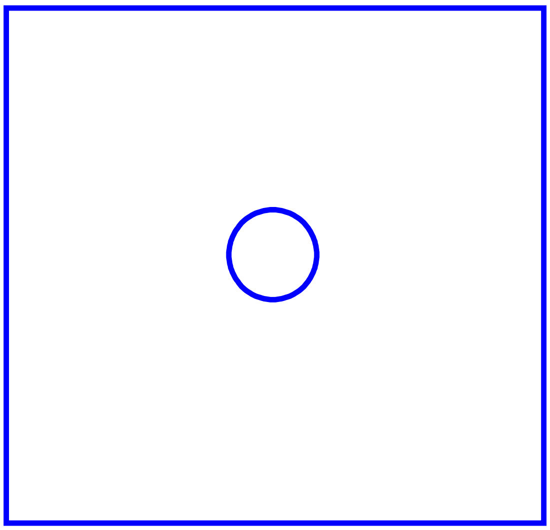


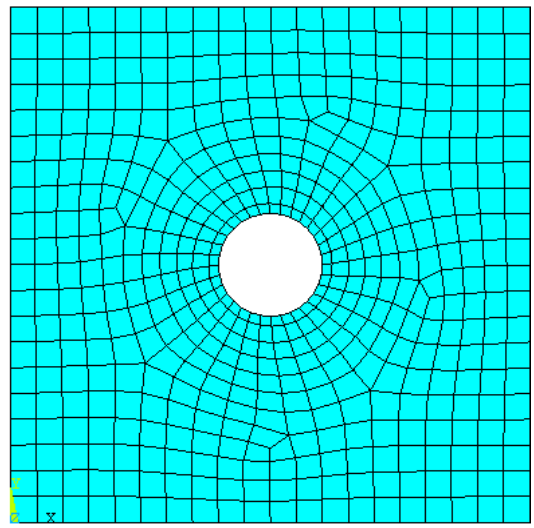
Figure 1.4. Simulating a can drop with dynamic FEM.

FEM in Structural Analysis (The Procedure in Using the FEM)

- Divide structure into pieces (elements with nodes, Figure 1.5)
- Describe the behavior of the physical quantities on each element
- Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure
- Solve the system of equations involving unknown quantities at the nodes (for example, the displacements)
- Calculate desired quantities (for example, strains and stresses) at selected elements



(a)



(b)

Figure 1.5. (a) A plate with a hole; and (b) A FEM discretization (mesh).

Computer Implementations

A typical FEM software has the following three key components:

- Preprocessor (used to build FE models, apply loads and constraints)
- FEA solver (assemble and solve the FEM system of equations)
- Postprocessor (sort and display the results)

The computer graphical-user interface (GUI) of the ANSYS software is shown in Figure 1.6. Other FEM packages have similar interfaces.

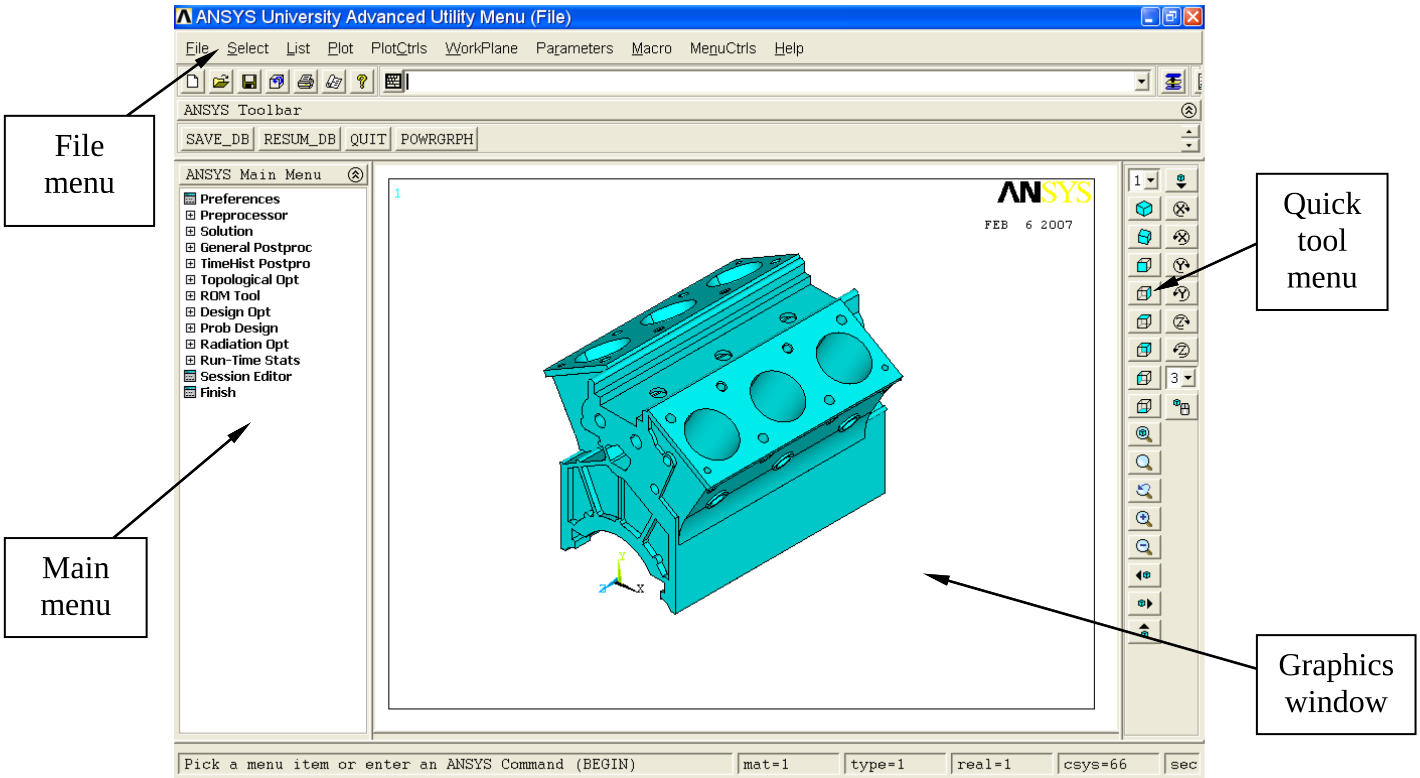


Figure 1.6. GUI of the ANSYS software.

Available Commercial FEM Software Packages

There are many commercial FEM packages, or CAD/CAE software in general, available for conducting FEA in nearly all fields of engineering. The following is only a short list:

- *ANSYS* (General purpose, PCs and workstations)
- *UG/NX* (Complete CAD/CAM/CAE package)
- *NASTRAN* (General purpose FEA on mainframes and PCs)
- *ABAQUS* (Nonlinear and dynamic analyses)
- *COSMOS* (General purpose FEA)
- *ALGOR* (PCs and workstations)
- *PATRAN* (Pre/Post Processor)
- *HyperWorks/HyperMesh* (Pre/Post Processor)
- *Dyna-3D* (Crash/impact analysis)
- *Others*

Objectives of This Course

- Understand the fundamental ideas of the FEM
- Know the behavior and usage of each type of elements covered in this course
- Be able to prepare a suitable FE model for a given problem
- Can interpret and evaluate the quality of the results (know the physics of the problems)
- Be aware of the limitations of the FEM (do not misuse the FEM)

II. Review of Matrix Algebra

Linear System of Algebraic Equations

$$\begin{aligned} &a_{11}x_1 + a_{12}x_2 + \ldots + a_{1n}x_n = b_1 \\ &a_{21}x_1 + a_{22}x_2 + \ldots + a_{2n}x_n = b_2 \\ &\ldots\ldots. \\ &a_{n1}x_1 + a_{n2}x_2 + \ldots + a_{nn}x_n = b_n \end{aligned} \tag{1.1}$$

where x_1, x_2, ..., x_n are the unknowns.

In *matrix form*:

$$\mathbf{Ax} = \mathbf{b} \tag{1.2}$$

where

$$\mathbf{A} = [a_{ij}] = \begin{bmatrix} a_{11} & a_{12} & \ldots & a_{1n} \\ a_{21} & a_{22} & \ldots & a_{2n} \\ \ldots & \ldots & \ldots & \ldots \\ a_{n1} & a_{n2} & \ldots & a_{nn} \end{bmatrix}, \quad \mathbf{x} = \{x_i\} = \begin{Bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{Bmatrix}, \quad \mathbf{b} = \{b_i\} = \begin{Bmatrix} b_1 \\ b_2 \\ \vdots \\ b_n \end{Bmatrix} \tag{1.3}$$

A is called a $n \times n$ (square) matrix, and **x** and **b** are (column) vectors with dimension n.

Row and Column Vectors

$$\mathbf{v} = [v_1 \quad v_2 \quad v_3] \quad \mathbf{w} = \begin{Bmatrix} w_1 \\ w_2 \\ w_3 \end{Bmatrix} \tag{1.4}$$

Matrix Addition and Subtraction

For two matrices **A** and **B**, both of the *same size* ($m \times n$), the addition and subtraction are defined by

$$
\begin{aligned}
\mathbf{C} &= \mathbf{A} + \mathbf{B} \quad \text{with} \quad c_{ij} = a_{ij} + b_{ij} \\
\mathbf{D} &= \mathbf{A} - \mathbf{B} \quad \text{with} \quad d_{ij} = a_{ij} - b_{ij}
\end{aligned} \tag{1.5}
$$

Scalar Multiplication

$$\lambda \mathbf{A} = \left[\lambda a_{ij} \right] \tag{1.6}$$

Matrix Multiplication

For two matrices **A** (of size $l \times m$) and **B** (of size $m \times n$), the product of **AB** is defined by

$$\mathbf{C} = \mathbf{AB} \quad \text{with } c_{ij} = \sum_{k=1}^{m} a_{ik} b_{kj} \tag{1.7}$$

where $i = 1, 2, ..., l$; $j = 1, 2, ..., n$.

Note that, in general, $\mathbf{AB} \neq \mathbf{BA}$, but $(\mathbf{AB})\mathbf{C} = \mathbf{A}(\mathbf{BC})$ (associative).

Transpose of a Matrix

If $\mathbf{A} = [a_{ij}]$, then the transpose of **A** is

$$\mathbf{A}^T = \left[a_{ji} \right] \tag{1.8}$$

Notice that

$$(\mathbf{AB})^T = \mathbf{B}^T \mathbf{A}^T \tag{1.9}$$

Symmetric Matrix

A *square* ($n \times n$) matrix **A** is called symmetric, if

$$\mathbf{A} = \mathbf{A}^T \qquad \text{or} \qquad a_{ij} = a_{ji} \tag{1.10}$$

Unit (Identity) Matrix

$$\mathbf{I} = \begin{bmatrix} 1 & 0 & ... & 0 \\ 0 & 1 & ... & 0 \\ ... & ... & ... & ... \\ 0 & 0 & ... & 1 \end{bmatrix} \tag{1.11}$$

Note that $\mathbf{AI} = \mathbf{A}$, $\mathbf{Ix} = \mathbf{x}$.

Determinant of a Matrix

The determinant of *square* matrix **A** is a scalar number denoted by det **A** or |**A**|. For 2×2 and 3×3 matrices, their determinants are given by

$$\det \begin{bmatrix} a & b \\ c & d \end{bmatrix} = ad - bc \tag{1.12}$$

and

$$\det\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} = a_{11}a_{22}a_{33} + a_{12}a_{23}a_{31} + a_{21}a_{32}a_{13} \\ - a_{13}a_{22}a_{31} - a_{12}a_{21}a_{33} - a_{23}a_{32}a_{11} \tag{1.13}$$

Singular Matrix

A *square* matrix **A** is *singular* if det **A** = 0, which indicates problems in the system (nonunique solutions, degeneracy, etc.)

Matrix Inversion

For a *square* and *nonsingular* matrix **A** ($\det \mathbf{A} \neq 0$), its *inverse* $\mathbf{A}^{-1}$ is constructed in such a way that

$$\mathbf{A}\mathbf{A}^{-1} = \mathbf{A}^{-1}\mathbf{A} = \mathbf{I} \tag{1.14}$$

The *cofactor matrix* **C** of matrix **A** is defined by

$$C_{ij} = (-1)^{i+j} M_{ij} \tag{1.15}$$

where M_{ij} is the determinant of the smaller matrix obtained by eliminating the ith row and jth column of **A**.

Thus, the inverse of **A** can be determined by

$$\mathbf{A}^{-1} = \frac{1}{\det \mathbf{A}} \mathbf{C}^T \tag{1.16}$$

We can show that $(\mathbf{AB})^{-1} = \mathbf{B}^{-1}\mathbf{A}^{-1}$.

Examples:

(A) $$\begin{bmatrix} a & b \\ c & d \end{bmatrix}^{-1} = \frac{1}{(ad - bc)}\begin{bmatrix} d & -b \\ -c & a \end{bmatrix}.$$

Checking:

$$\begin{bmatrix} a & b \\ c & d \end{bmatrix}^{-1}\begin{bmatrix} a & b \\ c & d \end{bmatrix} = \frac{1}{(ad - bc)}\begin{bmatrix} d & -b \\ -c & a \end{bmatrix}\begin{bmatrix} a & b \\ c & d \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}.$$

(B) $$\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 2 \end{bmatrix}^{-1} = \frac{1}{(4-2-1)}\begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix}^T = \begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix}.$$

Checking:

$$\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 2 \end{bmatrix}\begin{bmatrix} 3 & 2 & 1 \\ 2 & 2 & 1 \\ 1 & 1 & 1 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}.$$

If det **A** = 0 (i.e., **A** is *singular*), $\mathbf{A}^{-1}$ does not exist.

The solution of the linear system of equations (Eq. (1.1)) can be expressed as (assuming the coefficient matrix **A** is nonsingular)

$$\mathbf{x} = \mathbf{A}^{-1}\mathbf{b}$$

Thus, the main task in solving a linear system is to find the inverse of the coefficient matrix.

Solution Techniques for Linear Systems of Equations

- Gauss elimination methods
- Iterative methods

We will briefly review the two methods in Chapter 4.

Positive Definite Matrix

A *square* ($n \times n$) matrix **A** is said to be *positive definite*, if for all nonzero vector **x** of dimension n,

$$\mathbf{x}^T\mathbf{A}\mathbf{x} > 0$$

Note that positive definite matrices are nonsingular. Later on we will see that all stiffness matrices are positive definite and the above condition is a statement that the strain energy in a structure should be positive if the structure is constrained and the stiffness matrix is nonsingular.

Differentiation and Integration of a Matrix

Let $\mathbf{A}(t) = \left[a_{ij}(t)\right]$, then the differentiation is defined by

$$\frac{d}{dt}\mathbf{A}(t) = \left[\frac{da_{ij}(t)}{dt}\right] \tag{1.17}$$

and the integration by

$$\int \mathbf{A}(t)dt = \left[\int a_{ij}(t)dt\right] \tag{1.18}$$

Types of Finite Elements

We are now ready to study the various finite elements. All the elements developed in the FEM can be categorized into the following three types according to their geometries.

1-D (Line) Elements:

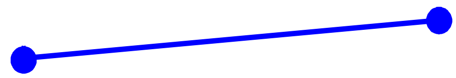


Figure 1.6. 1-D elements: Springs, trusses, beams, pipes, etc.

2-D (Plane) Elements:

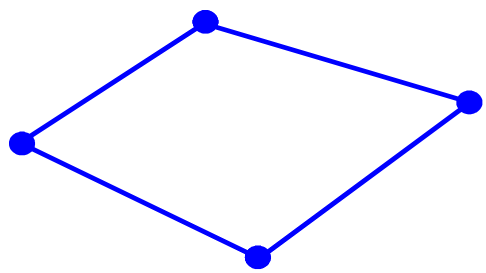


Figure 1.7. 2-D elements: Membranes, plates, shells, etc.

3-D (Solid) Elements:

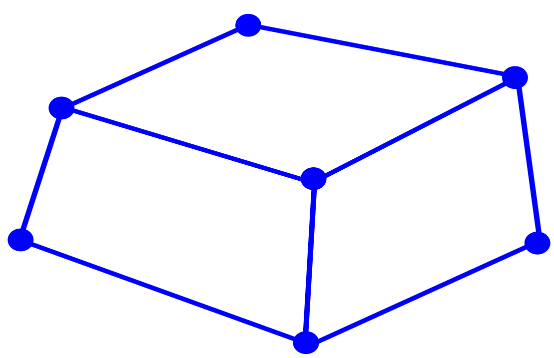


Figure 1.8. 3-D elements for 3-D fields (temperature, displacement, stress, velocity, etc.).

We will start with the 1-D spring element as an example to study the basic concept and ingredients in the FEM.

III. Spring Element

"Everything important is simple!"

One Spring Element

We first study a single spring element (Figure 1.9) and then a system of spring elements.

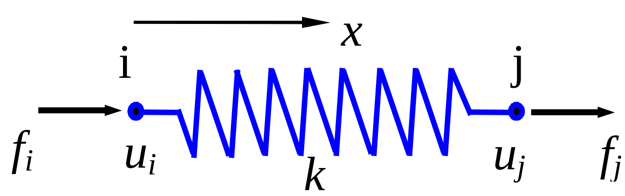


Figure 1.9. One spring element.

Two nodes:	i, j
Nodal displacements:	u_i, u_j (m, mm)
Nodal forces:	f_i, f_j (Newton)
Spring constant (stiffness):	k (N/m, N/mm)

Relationship between spring force F and elongation Δ is shown in Figure 1.10.

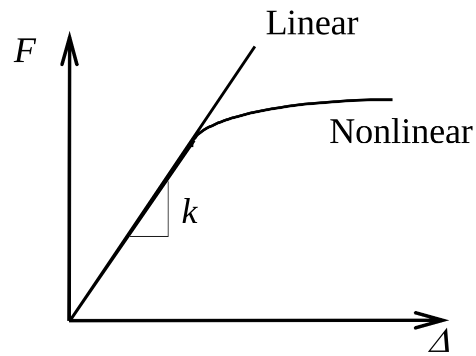


Figure 1.10. Force-displacement relation in a spring.

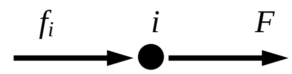
We only consider *linear* problems in this introductory course. In the linear portion of the curve shown in Figure 1.10, we have

$$F = k\Delta, \qquad \text{with} \quad \Delta = u_j - u_i. \tag{1.19}$$

where $k = F/\Delta$ (> 0) is the stiffness of the spring (the force needed to produce a unit stretch).

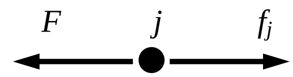
Consider the equilibrium of forces for the spring. At node i, we have

$$f_i = -F = -k(u_j - u_i) = ku_i - ku_j$$



and at node j,

$$f_j = F = k(u_j - u_i) = -ku_i + ku_j$$



In matrix form,

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix} \tag{1.20}$$

or,

$$\mathbf{ku} = \mathbf{f} \tag{1.21}$$

where

k = (element) stiffness matrix

u = (element) nodal displacement vector

f = (element) nodal force vector

From the derivation, we see that the first equation in (1.20) represents the equilibrium of forces at node i, while the second equation in (1.20) that of forces at node j. Note also that **k** is symmetric. Is **k** singular or nonsingular? That is, can we solve the equation in (1.20)? If not, why?

Spring System

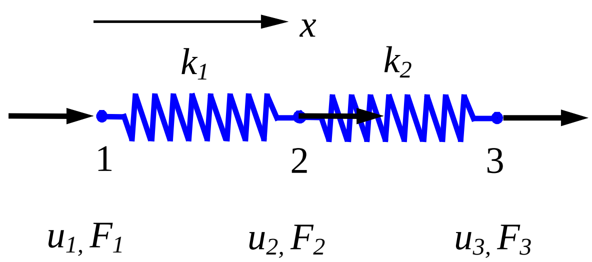


Figure 1.11. A system of two spring elements.

For a system of multiple spring elements, we first write down the stiffness equation for each spring and then "assemble" them together to form the stiffness equation for the whole system. For example, for the two-spring system shown in Figure 1.11, we proceed as follows:

For element 1, we have

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \end{Bmatrix} \tag{1.22}$$

and for element 2,

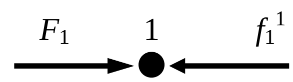
$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^2 \\ f_2^2 \end{Bmatrix} \tag{1.23}$$

where f_i^m is the (internal) force acting on *local* node *i* of element *m* (i = 1, 2).

Assemble the stiffness matrix for the whole system:

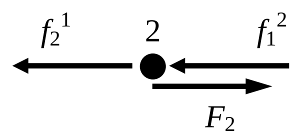
Consider the equilibrium of forces at node 1,

$$F_1 = f_1^1$$



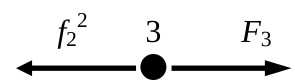
at node 2,

$$F_2 = f_2^1 + f_1^2$$



and at node 3,

$$F_3 = f_2^2$$



Using (1.22) and (1.23), we obtain

$$\begin{aligned} F_1 &= k_1 u_1 - k_1 u_2 \\ F_2 &= -k_1 u_1 + (k_1 + k_2) u_2 - k_2 u_3 \\ F_3 &= -k_2 u_2 + k_2 u_3 \end{aligned}$$

In matrix form, we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \tag{1.24}$$

or

$$\mathbf{K}\mathbf{u} = \mathbf{F} \tag{1.25}$$

in which, **K** is the stiffness matrix (structure matrix) for the entire spring system.

An alternative way of assembling the whole stiffness matrix:

"Enlarging" the stiffness matrices for elements 1 and 2, we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \\ 0 \end{Bmatrix},$$

and

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_1^2 \\ f_2^2 \end{Bmatrix}$$

Adding the two matrix equations (i.e., using *superposition*), we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 + f_1^2 \\ f_2^2 \end{Bmatrix}$$

This is the same equation we derived by using the force equilibrium concept.

Energy approach:

We can also obtain the result using an energy method, for example, the principle of minimum potential energy. In fact, the energy approach is more general and considered the foundation of the FEM. To proceed, we consider the strain energy U stored in the spring system shown in Figure 1.11.

$$U = \frac{1}{2}k_1\Delta_1{}^2 + \frac{1}{2}k_2\Delta_2{}^2 = \frac{1}{2}\Delta_1{}^T k_1 \Delta_1 + \frac{1}{2}\Delta_2{}^T k_2 \Delta_2$$

However,

$$\Delta_1 = u_2 - u_1 = \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}, \quad \Delta_2 = u_3 - u_2 = \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

We have

$$U = \frac{1}{2}\begin{bmatrix} u_1 & u_2 \end{bmatrix} \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} + \frac{1}{2}\begin{bmatrix} u_2 & u_3 \end{bmatrix} \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \text{(enlarging…)}$$

$$= \frac{1}{2}\begin{bmatrix} u_1 & u_2 & u_3 \end{bmatrix}\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \tag{1.26}$$

The potential of the external forces is

$$\Omega = -F_1u_1 - F_2u_2 - F_3u_3 = -\begin{bmatrix} u_1 & u_2 & u_3 \end{bmatrix}\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \tag{1.27}$$

Thus, the total potential energy of the system is

$$\Pi = U + \Omega = \frac{1}{2}\begin{bmatrix} u_1 & u_2 & u_3 \end{bmatrix}\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} - \begin{bmatrix} u_1 & u_2 & u_3 \end{bmatrix}\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \tag{1.28}$$

which is a function of the three nodal displacements (u_1, u_2, u_3). According to the principle of minimum potential energy, for a system to be in equilibrium, the total potential energy must be minimum, that is, $d\Pi = 0$, or equivalently,

$$\frac{\partial \Pi}{\partial u_1} = 0, \qquad \frac{\partial \Pi}{\partial u_2} = 0, \qquad \frac{\partial \Pi}{\partial u_3} = 0, \tag{1.29}$$

which yield the same three equations as in (1.24).

Boundary and load conditions:

Assuming that node 1 is fixed, and same force P is applied at node 2 and node 3, that is

$$u_1 = 0 \quad \text{and} \quad F_2 = F_3 = P$$

we have from Eq. (1.24)

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ P \end{Bmatrix}$$

which reduces to

$$\begin{bmatrix} k_1+k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P \\ P \end{Bmatrix}$$

and

$$F_1 = -k_1 u_2$$

Unknowns are

$$\mathbf{U} = \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$ and the reaction force F_1 (if desired).

Solving the equations, we obtain the displacements

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 2P/k_1 \\ 2P/k_1 + P/k_2 \end{Bmatrix}$$

and the reaction force

$$F_1 = -2P$$

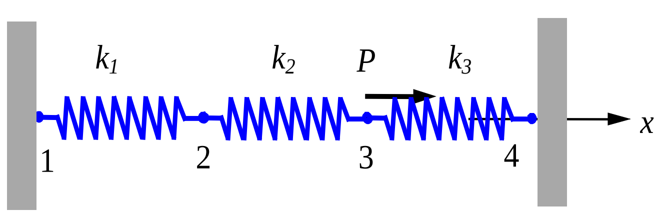
Checking the Results

- Deformed shape of the structure
- Equilibrium of the external forces
- Order of magnitudes of the obtained values

Notes About the Spring Elements

- Spring elements are only suitable for stiffness analysis
- They are not suitable for stress analysis of the spring itself
- There are spring elements with stiffness in the lateral direction, spring elements for torsion, etc.

Example 1.1



Given: For the spring system shown above,

$$k_1 = 100 \text{ N/mm}, \; k_2 = 200 \text{ N/mm}, \; k_3 = 100 \text{ N/mm}$$
$$P = 500 \text{ N}, \; \mathrm{u}_1 = u_4 = 0$$

Find: (a) the global stiffness matrix

(b) displacements of nodes 2 and 3

(c) the reaction forces at nodes 1 and 4

(d) the force in the spring 2

Solution:

(a) The element stiffness matrices are (make sure to put proper unit after each number)

$$\mathbf{k}_1 = \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \quad \text{(N/mm)}$$

$$\mathbf{k}_2 = \begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix} \quad \text{(N/mm)}$$

$$\mathbf{k}_3 = \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \quad \text{(N/mm)}$$

Applying the superposition concept, we obtain the global stiffness matrix for the spring system

$$\begin{array}{c} \\ \mathbf{K} = \end{array} \begin{array}{c} \begin{array}{cccc} u_1 & u_2 & u_3 & u_4 \end{array} \\ \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 100+200 & -200 & 0 \\ 0 & -200 & 200+100 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix} \end{array}$$

or

$$\mathbf{K} = \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 300 & -200 & 0 \\ 0 & -200 & 300 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix}$$

which is *symmetric* and *banded*.

Equilibrium (FE) equation for the whole system is

$$\begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 300 & -200 & 0 \\ 0 & -200 & 300 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}$$

(b) Applying the BCs $u_1 = u_4 = 0$, $F_2 = 0$ and $F_3 = P$, and "deleting" the 1st and 4th rows and columns, we have

$$\begin{bmatrix} 300 & -200 \\ -200 & 300 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ P \end{Bmatrix}$$

Solving this equation, we obtain

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P/250 \\ 3P/500 \end{Bmatrix} = \begin{Bmatrix} 2 \\ 3 \end{Bmatrix} (\text{mm})$$

(c) From the 1st and 4th equations system FE equation, we obtain the reaction forces

$$F_1 = -100 u_2 = -200 (\text{N})$$

$$F_4 = -100 u_3 = -300 (\text{N})$$

(d) The FE equation for spring (element) 2 is

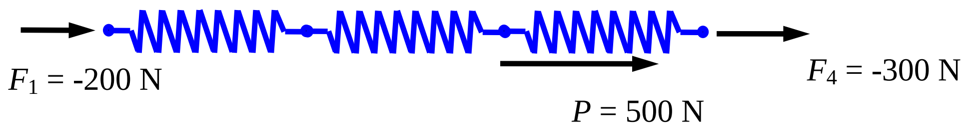
$$\begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix}$$

Here $i = 2$, $j = 3$ for element 2. Thus we can calculate the spring force as

$$F = f_j = -f_i = \begin{bmatrix} -200 & 200 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} -200 & 200 \end{bmatrix} \begin{Bmatrix} 2 \\ 3 \end{Bmatrix} = 200 (\text{N})$$

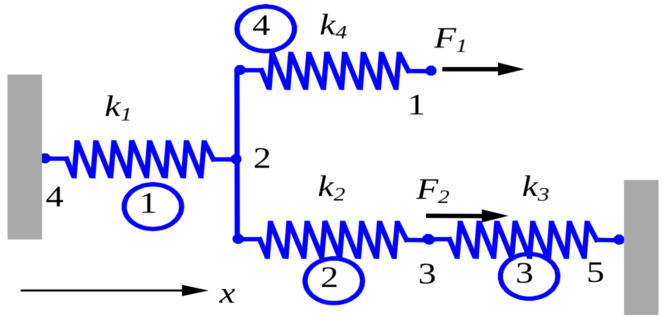
Check the results:

Draw the free-body diagram (FBD) of the system and consider the equilibrium of the forces.



Equilibrium of the forces is satisfied!

Example 1.2



Problem: For the spring system with arbitrarily numbered nodes and elements, as shown above, find the global stiffness matrix.

Solution: First we construct the following *element connectivity table*

Element Connectivity Table

Element	Node i (1)	Node j (2)
1	4	2
2	2	3
3	3	5
4	2	1

which specifies the *global* node numbers corresponding to the *local* node numbers for each element.

Then we write the element stiffness matrix for each element

$$\mathbf{k}_1 = \begin{array}{c} \begin{array}{cc} u_4 & u_2 \end{array} \\ \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \end{array}, \quad \mathbf{k}_2 = \begin{array}{c} \begin{array}{cc} u_2 & u_3 \end{array} \\ \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \end{array}, \quad \mathbf{k}_3 = \begin{array}{c} \begin{array}{cc} u_3 & u_5 \end{array} \\ \begin{bmatrix} k_3 & -k_3 \\ -k_3 & k_3 \end{bmatrix} \end{array}, \quad \mathbf{k}_4 = \begin{array}{c} \begin{array}{cc} u_2 & u_1 \end{array} \\ \begin{bmatrix} k_4 & -k_4 \\ -k_4 & k_4 \end{bmatrix} \end{array}$$

Finally, applying the superposition method, we obtain the global stiffness matrix as follows

$$\mathbf{K} = \begin{array}{c} \begin{array}{ccccc} u_1 & u_2 & u_3 & u_4 & u_5 \end{array} \\ \left[\begin{array}{c|c|c|c|c} k_4 & -k_4 & 0 & 0 & 0 \\ \hline -k_4 & k_1+k_2+k_4 & -k_2 & -k_1 & 0 \\ \hline 0 & -k_2 & k_2+k_3 & 0 & -k_3 \\ \hline 0 & -k_1 & 0 & k_1 & 0 \\ \hline 0 & 0 & -k_3 & 0 & k_3 \end{array} \right] \end{array}$$

The matrix is *symmetric*, *banded*, but *singular*.

IV. Summary

In this chapter, the basic concepts in the finite element method are introduced. The spring element is used as an example to show how to establish the element stiffness matrices, to assemble the finite element equations for a system from element stiffness matrices, and to solve the FE equations. Verifying the FE results is emphasized. The concepts and procedures introduced in this chapter are very simple and yet very important for studying the finite element analyses of other problems.

V. Problems

***Problem 1*.** Answer the following questions briefly:

(a) What is the physical meaning of the FE equations (for either an element or the whole structure)?

(b) What is the procedure in using the FEM?

***Problem 2*.** For the following given matrix and vector

$$\mathbf{A} = \begin{bmatrix} 1 & -2 & 6 \\ 2 & 2 & 3 \\ -1 & 3 & 0 \end{bmatrix}, \qquad \mathbf{b} = \begin{Bmatrix} 0 \\ 3 \\ 2 \end{Bmatrix}$$

Find:

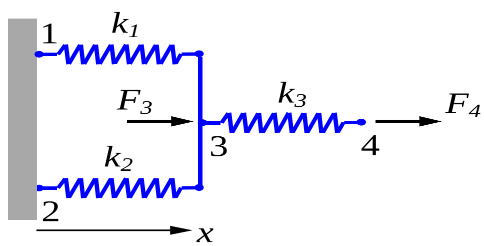
(a) Determinant det $\mathbf{A}$;

(b) Inverse $\mathbf{A}^{-1}$;

(c) Solution of the equation $\mathbf{Ax} = \mathbf{b}$;

(d) Value of the quadratic form $\mathbf{x}^T\mathbf{Ax}$.

***Problem 3*.** A spring system is shown below



Given: $k_1 = 80$ N/mm, $k_2 = 100$ N/mm, $k_3 = 160$ N/mm, $F_3 = 200$ N, $F_4 = 100$ N, and nodes 1 and 2 are fixed;

Find:

(a) Global stiffness matrix;

(b) Displacements of nodes 3 and 4;

(c) Reaction forces at nodes 1 and 2;

(d) Forces in springs 1 and 2.

Chapter 2. Bar and Beam Elements

I. Linear Static Analysis

Most structural analysis problems can be treated as *linear static* problems, based on the following assumptions

1. *Small deformations* (loading pattern is not changed due to the deformed shape)
2. *Elastic materials* (no plasticity or failures)
3. *Static loads* (the load is applied to the structure in a slow or steady fashion)

Linear analysis can provide most of the information about the behavior of a structure, and can be a good approximation for many analyses. It is also the bases of nonlinear analysis in most of the cases.

II. Bar Element

Consider a uniform prismatic bar:

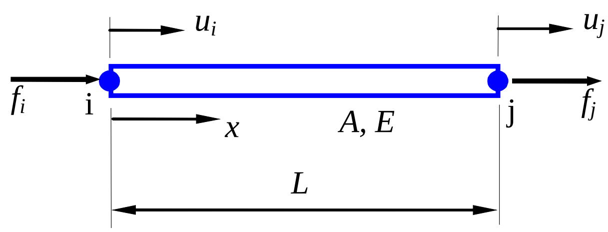


Figure 2.1. Notations for a bar element.

L, A, E	length, cross-sectional area, and elastic modulus of the beam, respectively
u, ε, σ	displacement, strain, and stress, respectively (all functions of x)

Strain-displacement relation:

$$\varepsilon = \frac{du}{dx} \tag{2.1}$$

Stress-strain relation:

$$\sigma = E\varepsilon \tag{2.2}$$

Equilibrium equation:

$$\frac{d\sigma}{dx} + f = 0 \tag{2.3}$$

where f is the body force (force per volume, such as gravitational and magnetic forces) inside the bar.

Stiffness Matrix --- Direct Method

Assuming that the displacement u is *varying linearly* along the axis of the bar, that is, in terms of the two nodal values u_i and u_j, we can write (derive this)

$$u(x) = \left(1 - \frac{x}{L}\right)u_i + \frac{x}{L}u_j \tag{2.4}$$

we have

$$\varepsilon = \frac{u_j - u_i}{L} = \frac{\Delta}{L} \qquad (\Delta \text{ = elongation}) \tag{2.5}$$

$$\sigma = E\varepsilon = \frac{E\Delta}{L} \tag{2.6}$$

We also have

$$\sigma = \frac{F}{A} \qquad (F \text{ = force in bar}) \tag{2.7}$$

Thus, (2.6) and (2.7) lead to

$$F = \frac{EA}{L}\Delta = k\Delta$$

where $k = \dfrac{EA}{L}$ is the stiffness of the bar. That is, the bar behaves *like a spring* in this case and we conclude that the element stiffness matrix is

$$\mathbf{k} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

or

$$\mathbf{k} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{2.8}$$

This can be verified by considering the equilibrium of the forces at the two nodes.

Element equilibrium equation is

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix} \tag{2.9}$$

Degree of Freedom (DOF): Number of components of the displacement vector at a node. For 1-D bar element: one DOF at each node.

Physical Meaning of the Coefficients in **k**: The jth column of **k** (here j = 1 or 2) represents the forces applied to the bar to maintain a deformed shape with unit displacement at node j and zero displacement at the other node.

Stiffness Matrix --- A Formal Approach

We derive the same stiffness matrix for the bar using a formal approach which can be applied to many other more complicated situations.

Define two *linear shape functions* as follows

$$N_i(\xi) = 1 - \xi, \qquad N_j(\xi) = \xi \tag{2.10}$$

where

$$\xi = \frac{x}{L}, \qquad 0 \le \xi \le 1 \tag{2.11}$$

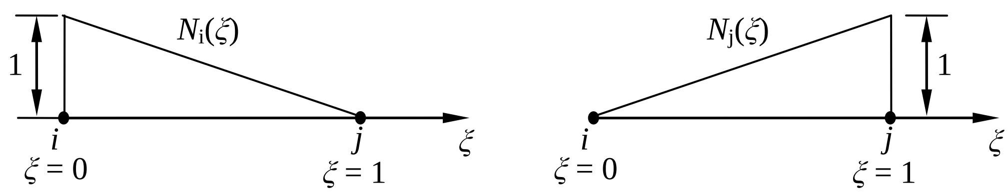


Figure 2.2. The shape of the shape functions.

From (2.4) we can write the displacement as

$$u(x) = u(\xi) = N_i(\xi)u_i + N_j(\xi)u_j$$

or

$$u = \begin{bmatrix} N_i & N_j \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \mathbf{Nu} \tag{2.12}$$

Strain is given by (2.1) and (2.12) as

$$\varepsilon = \frac{du}{dx} = \left[\frac{d}{dx}\mathbf{N}\right]\mathbf{u} = \mathbf{Bu} \tag{2.13}$$

where **B** is the element *strain-displacement matrix*, which is

$$\mathbf{B} = \frac{d}{dx}\begin{bmatrix} N_i(\xi) & N_j(\xi) \end{bmatrix} = \frac{d}{d\xi}\begin{bmatrix} N_i(\xi) & N_j(\xi) \end{bmatrix} \bullet \frac{d\xi}{dx}$$

that is,

$$\mathbf{B} = \begin{bmatrix} -1/L & 1/L \end{bmatrix} \tag{2.14}$$

Stress can be written as

$$\sigma = E\varepsilon = E\mathbf{Bu} \tag{2.15}$$

Consider the *strain energy* stored in the bar

$$\begin{aligned} U &= \frac{1}{2}\int_V \sigma^{\mathbf{T}} \varepsilon dV = \frac{1}{2}\int_V \left(\mathbf{u}^{\mathbf{T}}\mathbf{B}^{\mathbf{T}} E\mathbf{Bu}\right) dV \\ &= \frac{1}{2}\mathbf{u}^{\mathbf{T}}\left[\int_V \left(\mathbf{B}^{\mathbf{T}} E\mathbf{B}\right) dV\right]\mathbf{u} \end{aligned} \tag{2.16}$$

where (2.13) and (2.15) have been used.

The *potential* of the external forces is written as (this is by definition, and remember the negative sign)

$$\Omega = -f_i u_i - f_j u_j = -\mathbf{u}^{\mathbf{T}}\mathbf{f} \tag{2.17}$$

The total potential of the system is

$$\Pi = U + \Omega$$

which yields by using (2.16) and (2.17)

$$\Pi = \frac{1}{2}\mathbf{u}^{\mathbf{T}}\left[\int_V \left(\mathbf{B}^{\mathbf{T}} E\mathbf{B}\right) dV\right]\mathbf{u} - \mathbf{u}^{\mathbf{T}}\mathbf{f} \tag{2.18}$$

Setting $d\Pi = 0$ by the principle of minimum potential energy, we obtain (verify this)

$$\left[\int_V \left(\mathbf{B}^{\mathbf{T}} E\mathbf{B}\right) dV\right]\mathbf{u} = \mathbf{f}$$

or

$$\mathbf{ku} = \mathbf{f} \tag{2.19}$$

where

$$\mathbf{k} = \int_V \left(\mathbf{B}^{\mathbf{T}} E\mathbf{B}\right) dV \tag{2.20}$$

is the *element stiffness matrix*.

Expression (2.20) is a general result which can be used for the construction of other types of elements.

Now, we evaluate (2.20) for the bar element by using (2.14)

$$\mathbf{k} = \int_0^L \begin{Bmatrix} -1/L \\ 1/L \end{Bmatrix} E\begin{bmatrix} -1/L & 1/L \end{bmatrix} A dx = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

which is the same as we derived earlier using the direct method.

Note that from (2.16) and (2.20), the strain energy in the element can be written as

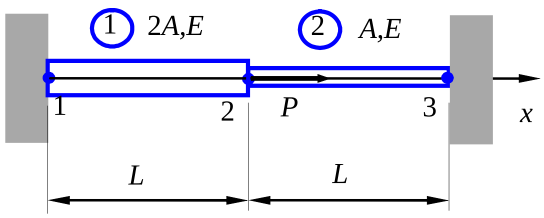
$$U = \frac{1}{2}\mathbf{u}^{\mathrm{T}}\mathbf{k}\mathbf{u} \tag{2.21}$$

In the future, once we obtain an expression like (2.16), we can immediately recognize that the matrix within the square bracket is the stiffness matrix. Recall that for a spring, the strain energy can be written as

$$U = \frac{1}{2}k\Delta^2 = \frac{1}{2}\Delta^T k\Delta$$

Thus result (2.21) goes back to the simple spring case again.

Example 2.1



Problem: Find the stresses in the two-bar assembly which is loaded with force *P*, and constrained at the two ends, as shown in the above figure.

Solution: Use two 1-D bar elements.

For element 1,

$$\mathbf{k}_1 = \frac{2EA}{L}\begin{array}{c} \begin{matrix} u_1 & \quad u_2 \end{matrix} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}$$

For element 2,

$$\mathbf{k}_2 = \frac{EA}{L}\begin{array}{c} \begin{matrix} u_2 & \quad u_3 \end{matrix} \\ \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}$$

Imagine a frictionless pin at node 2, which connects the two elements. We can assemble the global FE equation as follows

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Load and boundary conditions (BCs) are

$$u_1 = u_3 = 0, \qquad\qquad F_2 = P$$

FE equation becomes

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix}$$

"Deleting" the 1st row and column, and the 3rd row and column, we obtain

$$\frac{EA}{L}[3]\{u_2\} = \{P\}$$

Thus,

$$u_2 = \frac{PL}{3EA}$$

and

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{PL}{3EA}\begin{Bmatrix} 0 \\ 1 \\ 0 \end{Bmatrix}$$

Stress in element 1 is

$$\sigma_1 = E\varepsilon_1 = E\mathbf{B}_1\mathbf{u}_1 = E\left[-1/L \quad 1/L\right]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

$$= E\frac{u_2 - u_1}{L} = \frac{E}{L}\left(\frac{PL}{3EA} - 0\right) = \frac{P}{3A}$$

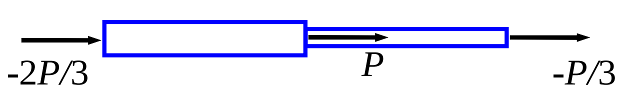
Similarly, stress in element 2 is

$$\sigma_2 = E\varepsilon_2 = E\mathbf{B}_2\mathbf{u}_2 = E\left[-1/L \quad 1/L\right]\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

$$= E\frac{u_3 - u_2}{L} = \frac{E}{L}\left(0 - \frac{PL}{3EA}\right) = -\frac{P}{3A}$$

which indicates that bar 2 is in compression.

***Check the results*:** Draw the FBD and check the equilibrium of the structures.

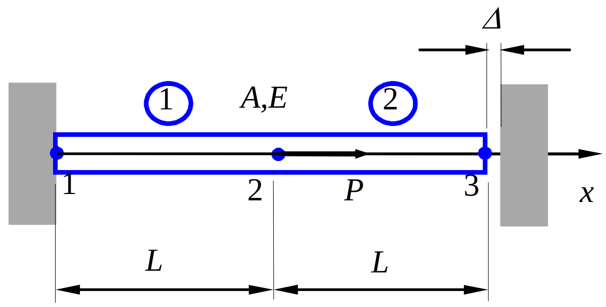


Notes:

- In this case, the calculated stresses in elements 1 and 2 are exact. It will not help if we further divide element 1 or 2 into smaller elements.

- For tapered bars, averaged values of the cross-sectional areas should be used for the elements.
- We need to find the displacements first in order to find the stresses, and thus this approach is called the *displacement based FEM*.

Example 2.2



Problem: Determine the support reaction forces at the two ends of the bar shown above, given the following

$$P = 6.0\times 10^4 \text{ N}, \quad E = 2.0\times 10^4 \text{ N/mm}^2,$$
$$A = 250\,\text{mm}^2, \quad L = 150\,\text{mm}, \quad \Delta = 1.2 \text{ mm}$$

Solution:

We first check to see if contact of the bar with the wall on the right will occur or not. To do this, we imagine the wall on the right is removed and calculate the displacement at the right end

$$\Delta_0 = \frac{PL}{EA} = \frac{(6.0\times 10^4)(150)}{(2.0\times 10^4)(250)} = 1.8\text{mm} > \Delta = 1.2\text{mm}$$

Thus, contact occurs and the wall on the right should be accounted for in the analysis.

The global FE equation is found to be

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

The load and boundary conditions are

$$F_2 = P = 6.0\times 10^4\text{N}$$
$$u_1 = 0, \qquad u_3 = \Delta = 1.2\text{mm}$$

FE equation becomes

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ \Delta \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix}$$

The 2nd equation gives

$$\frac{EA}{L}\begin{bmatrix} 2 & -1 \end{bmatrix}\begin{Bmatrix} u_2 \\ \Delta \end{Bmatrix} = \{P\}$$

that is,

$$\frac{EA}{L}[2]\{u_2\} = \left\{P + \frac{EA}{L}\Delta\right\}$$

Solving this, we obtain

$$u_2 = \frac{1}{2}\left(\frac{PL}{EA} + \Delta\right) = 1.5\text{mm}$$

and

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1.5 \\ 1.2 \end{Bmatrix} (\text{mm})$$

To calculate the support reaction forces, we apply the 1st and 3rd equations in the global FE equation.

The 1st equation gives

$$F_1 = \frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{EA}{L}(-u_2) = -5.0 \times 10^4 \text{ N}$$

and the 3rd equation gives,

$$F_3 = \frac{EA}{L}\begin{bmatrix} 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{EA}{L}(-u_2 + u_3) = -1.0 \times 10^4 \text{ N}$$

Check the results.!

Distributed Load

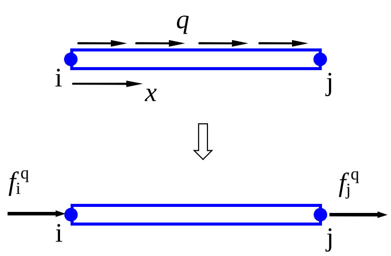


Figure 2.3. Conversion of a distributed load on one element.

Distributed axial load q (N/mm, N/m, lb/in) can be converted to two equivalent nodal forces using the shape functions. Consider the work done by the distributed load q,

$$W_q = \frac{1}{2}\int_0^L u(x)q(x)dx = \frac{1}{2}\int_0^L (\mathbf{Nu})^T q(x)dx = \frac{1}{2}\begin{bmatrix} u_i & u_j \end{bmatrix}\int_0^L \begin{bmatrix} N_i(x) \\ N_j(x) \end{bmatrix} q(x)dx$$
$$= \frac{1}{2}\mathbf{u}^T \int_0^L \mathbf{N}^T q(x)dx \tag{2.22}$$

The work done by the equivalent nodal forces are

$$W_{f_q} = \frac{1}{2} f_i^q u_i + \frac{1}{2} f_j^q u_j = \frac{1}{2}\mathbf{u}^T \mathbf{f}_q \tag{2.23}$$

Setting $W_q = W_{f_q}$ and using (2.22) and (2.23), we obtain the equivalent nodal force vector

$$\mathbf{f}_q = \begin{Bmatrix} f_i^q \\ f_j^q \end{Bmatrix} = \int_0^L \mathbf{N}^T q(x)dx = \int_0^L \begin{bmatrix} N_i(x) \\ N_j(x) \end{bmatrix} q(x)dx \tag{2.24}$$

which is valid for any distributions of q. If q is a *constant*, we have

$$\mathbf{f}_q = q\int_0^L \begin{bmatrix} 1 - x/L \\ x/L \end{bmatrix} dx = \begin{Bmatrix} qL/2 \\ qL/2 \end{Bmatrix} \tag{2.25}$$

In an assembly of bar elements, equivalent forces are added at each node as shown below.

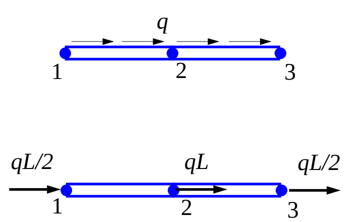


Figure 2.4. Conversion of a distributed load on two elements.

Bar Elements in 2-D and 3-D Spaces

2-D Case

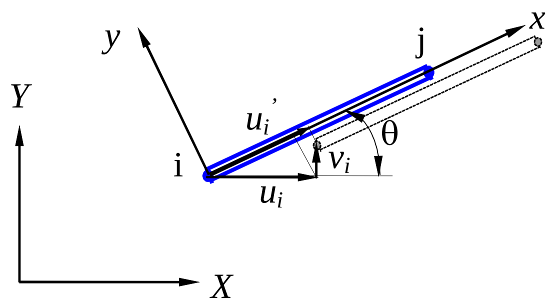


Figure 2.5. Local and global coordinates for a bar in 2-D space.

Local	*Global*
x, y	X, Y
u_i', v_i'	u_i, v_i
1 DOF at each node	2 DOFs at each node

Note that lateral displacement v_i' does not contribute to the stretch of the bar within the linear theory.

Transformation

$$u_i' = u_i \cos\theta + v_i \sin\theta = \begin{bmatrix} l & m \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

$$v_i' = -u_i \sin\theta + v_i \cos\theta = \begin{bmatrix} -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

where $l = \cos\theta, m = \sin\theta$.

In matrix form,

$$\begin{Bmatrix} u_i' \\ v_i' \end{Bmatrix} = \begin{bmatrix} l & m \\ -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix} \tag{2.26}$$

or,

$$\mathbf{u}_i' = \tilde{\mathbf{T}}\mathbf{u}_i$$

where the *transformation matrix*

$$\tilde{\mathbf{T}} = \begin{bmatrix} l & m \\ -m & l \end{bmatrix} \tag{2.27}$$

is *orthogonal*, that is, $\tilde{\mathbf{T}}^{-1} = \tilde{\mathbf{T}}^{T}$.

For the two nodes of the bar element, we have

$$\begin{Bmatrix} u_i' \\ v_i' \\ u_j' \\ v_j' \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ -m & l & 0 & 0 \\ 0 & 0 & l & m \\ 0 & 0 & -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix} \tag{2.28}$$

or,

$$\mathbf{u}' = \mathbf{T}\mathbf{u} \qquad \text{with} \qquad \mathbf{T} = \begin{bmatrix} \tilde{\mathbf{T}} & \mathbf{0} \\ \mathbf{0} & \tilde{\mathbf{T}} \end{bmatrix} \tag{2.29}$$

The nodal forces are transformed in the same way,

$$\mathbf{f}' = \mathbf{T}\mathbf{f} \tag{2.30}$$

Stiffness Matrix in the 2-D Space

In the local coordinate system, we have

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i' \\ u_j' \end{Bmatrix} = \begin{Bmatrix} f_i' \\ f_j' \end{Bmatrix}$$

Augmenting this equation, we write

$$\frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_i' \\ v_i' \\ u_j' \\ v_j' \end{Bmatrix} = \begin{Bmatrix} f_i' \\ 0 \\ f_j' \\ 0 \end{Bmatrix}$$

or,

$$\mathbf{k}'\mathbf{u}' = \mathbf{f}'$$

Using transformations given in (29) and (30), we obtain

$$\mathbf{k}'\mathbf{T}\mathbf{u} = \mathbf{T}\mathbf{f}$$

Multiplying both sides by $\mathbf{T}^T$ and noticing that $\mathbf{T}^T\mathbf{T} = \mathbf{I}$, we obtain

$$\mathbf{T}^T\mathbf{k}'\mathbf{T}\mathbf{u} = \mathbf{f} \tag{2.31}$$

Thus, the element stiffness matrix **k** in the global coordinate system is

$$\mathbf{k} = \mathbf{T}^T \mathbf{k}' \mathbf{T} \tag{2.32}$$

which is a 4×4 symmetric matrix.

Explicit form,

$$\mathbf{k} = \frac{EA}{L} \begin{array}{c} \begin{matrix} u_i & \quad v_i & \quad u_j & \quad v_j \end{matrix} \\ \begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix} \end{array} \tag{2.33}$$

Calculation of the *directional cosines* l and m:

$$l = \cos\theta = \frac{X_j - X_i}{L}, \quad m = \sin\theta = \frac{Y_j - Y_i}{L}$$

The structure stiffness matrix is assembled by using the element stiffness matrices in the usual way as in the 1-D case.

Element Stress

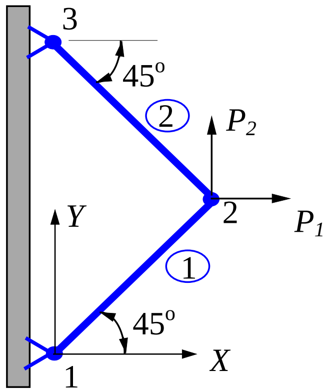
$$\sigma = E\varepsilon = E\mathbf{B}\begin{Bmatrix} u_i' \\ u_j' \end{Bmatrix} = E\begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}\begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

That is,

$$\sigma = \frac{E}{L}\begin{bmatrix} -l & -m & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix} \tag{2.34}$$

which can be used to evaluate the element stress once the nodal displacements are known.

Example 2.3



A simple plane truss is made of two identical bars (with E, A, and L), and loaded as shown in the above figure.

Find:

(a) displacement of node 2;

(b) stress in each bar.

Solution:

This simple structure is used here to demonstrate the FEA procedure using the bar element in 2-D space.

In local coordinate systems, we have

$$\mathbf{k}_1^{'} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \mathbf{k}_2^{'}$$

These two matrices cannot be assembled together, because they are in different coordinate systems. We need to convert them to global coordinate system OXY.

Element 1: $\theta = 45^o$, $l = m = \dfrac{\sqrt{2}}{2}$

Using formula (2.32) or (2.33), we obtain the stiffness matrix in the global system

$$\mathbf{k}_1 = \mathbf{T}_1^T \mathbf{k}_1^{'} \mathbf{T}_1 = \frac{EA}{2L}\begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_2 & v_2 \end{array} \\ \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \end{array}$$

Element 2: $\theta = 135^o$, $l = -\frac{\sqrt{2}}{2}, m = \frac{\sqrt{2}}{2}$

$$\mathbf{k}_2 = \mathbf{T}_2^T \mathbf{k}_2^{'} \mathbf{T}_2 = \frac{EA}{2L} \begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix} \end{array}$$

Assemble the structure FE equation,

$$\frac{EA}{2L} \begin{array}{c} \begin{array}{cccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & 1 & -1 & -1 & 0 & 0 \\ 1 & 1 & -1 & -1 & 0 & 0 \\ -1 & -1 & 2 & 0 & -1 & 1 \\ -1 & -1 & 0 & 2 & 1 & -1 \\ 0 & 0 & -1 & 1 & 1 & -1 \\ 0 & 0 & 1 & -1 & -1 & 1 \end{bmatrix} \end{array} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}$$

Load and boundary conditions (BC):

$$u_1 = v_1 = u_3 = v_3 = 0, \quad F_{2X} = P_1, F_{2Y} = P_2$$

Condensed FE equation,

$$\frac{EA}{2L} \begin{bmatrix} 2 & 0 \\ 0 & 2 \end{bmatrix} \begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

Solving this, we obtain the displacement of node 2,

$$\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \frac{L}{EA} \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

Using formula (2.34), we calculate the stresses in the two bars,

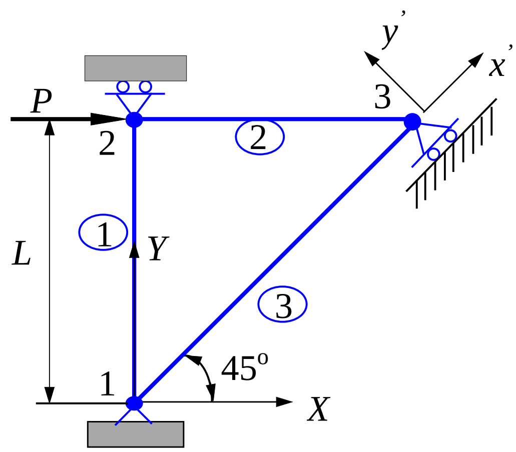
$$\sigma_1 = \frac{E}{L} \frac{\sqrt{2}}{2} \begin{bmatrix} -1 & -1 & 1 & 1 \end{bmatrix} \frac{L}{EA} \begin{Bmatrix} 0 \\ 0 \\ P_1 \\ P_2 \end{Bmatrix} = \frac{\sqrt{2}}{2A} \left(P_1 + P_2 \right)$$

$$\sigma_2 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix}1 & -1 & -1 & 1\end{bmatrix}\frac{L}{EA}\begin{Bmatrix}P_1\\P_2\\0\\0\end{Bmatrix} = \frac{\sqrt{2}}{2A}(P_1 - P_2)$$

***Check the results*:**

Check the equilibrium conditions, symmetry, antisymmetry, etc.

Example 2.4 (Multipoint Constraint)



For the plane truss shown above,

$P = 1000$ kN, $L = 1m$, $E = 210GPa$,

$A = 6.0\times10^{-4}m^2$ for elements 1 and 2,

$A = 6\sqrt{2}\times10^{-4}m^2$ for element 3.

Determine the displacements and reaction forces.

Solution:

We have an inclined roller at node 3, which needs special attention in the FE solution. We first assemble the global FE equation for the truss.

Element 1: $\theta = 90^o$, $l = 0, m = 1$

$$\mathbf{k}_1 = \frac{(210\times10^9)(6.0\times10^{-4})}{1}\begin{array}{c}\begin{matrix} u_1 & v_1 & u_2 & v_2 \end{matrix}\\ \begin{bmatrix}0 & 0 & 0 & 0\\0 & 1 & 0 & -1\\0 & 0 & 0 & 0\\0 & -1 & 0 & 1\end{bmatrix}\end{array} \text{ (N/m)}$$

Element 2: $\theta = 0^o$, $l = 1, m = 0$

$$\mathbf{k}_2 = \frac{(210\times10^9)(6.0\times10^{-4})}{1}\begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \text{(N/m)}$$

Element 3: $\theta = 45^o$, $l = \frac{1}{\sqrt{2}}, m = \frac{1}{\sqrt{2}}$

$$\mathbf{k}_3 = \frac{(210\times10^9)(6\sqrt{2}\times10^{-4})}{\sqrt{2}}\begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix} \end{array} \text{(N/m)}$$

The global FE equation is,

$$1260\times10^5\begin{bmatrix} 0.5 & 0.5 & 0 & 0 & -0.5 & -0.5 \\ & 1.5 & 0 & -1 & -0.5 & -0.5 \\ & & 1 & 0 & -1 & 0 \\ & & & 1 & 0 & 0 \\ & & & & 1.5 & 0.5 \\ Sym. & & & & & 0.5 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}$$

Load and boundary conditions (BCs):

$$u_1 = v_1 = v_2 = 0, \text{and } v_3^{'} = 0,$$
$$F_{2X} = P, \qquad F_{3x'} = 0.$$

From the transformation relation and the BC, we have

$$v_3^{'} = \begin{bmatrix} -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix}\begin{Bmatrix} u_3 \\ v_3 \end{Bmatrix} = \frac{\sqrt{2}}{2}(-u_3 + v_3) = 0,$$

that is,

$$u_3 - v_3 = 0$$

This is a *multipoint constraint* (MPC).

Similarly, we have a relation for the force at node 3,

$$F_{3x'} = \begin{bmatrix} \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix}\begin{Bmatrix} F_{3X} \\ F_{3Y} \end{Bmatrix} = \frac{\sqrt{2}}{2}(F_{3X} + F_{3Y}) = 0,$$

that is,

$$F_{3X} + F_{3Y} = 0$$

Applying the load and BC's in the structure FE equation by "deleting" the 1st, 2nd and 4th rows and columns, we have

$$1260 \times 10^5 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} P \\ F_{3X} \\ F_{3Y} \end{Bmatrix}$$

Further, from the MPC and the force relation at node 3, the equation becomes,

$$1260 \times 10^5 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P \\ F_{3X} \\ -F_{3X} \end{Bmatrix}$$

which is

$$1260 \times 10^5 \begin{bmatrix} 1 & -1 \\ -1 & 2 \\ 0 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P \\ F_{3X} \\ -F_{3X} \end{Bmatrix}$$

The 3rd equation yields,

$$F_{3X} = -1260 \times 10^5 u_3$$

Substituting this into the 2nd equation and rearranging, we have

$$1260 \times 10^5 \begin{bmatrix} 1 & -1 \\ -1 & 3 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P \\ 0 \end{Bmatrix}$$

Solving this, we obtain the displacements,

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{1}{2520 \times 10^5} \begin{Bmatrix} 3P \\ P \end{Bmatrix} = \begin{Bmatrix} 0.01191 \\ 0.003968 \end{Bmatrix} \text{(m)}$$

From the global FE equation, we can calculate the reaction forces,

$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = 1260 \times 10^5 \begin{bmatrix} 0 & -0.5 & -0.5 \\ 0 & -0.5 & -0.5 \\ 0 & 0 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} -500 \\ -500 \\ 0.0 \\ -500 \\ 500 \end{Bmatrix} \text{(kN)}$$

Check the results!

A general *multipoint constraint* (MPC) can be described as,

$$\sum_j A_j u_j = 0$$

where A_j's are constants and u_j's are nodal displacement components. In FE software, users only need to specify this relation to the software. The software will take care of the solution process.

3-D Case

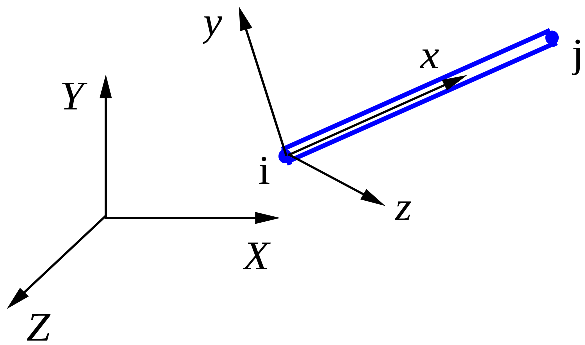


Figure 2.6. Local and global coordinates for a bar in 3-D space.

Local	*Global*
x, y, z $u_i^{'}, v_i^{'}, w_i^{'}$ 1 DOF at each node	X, Y, Z u_i, v_i, w_i 3 DOFs at each node

Element stiffness matrices are calculated in the local coordinate systems and then transformed into the global coordinate system (*X*, *Y*, *Z*) where they are assembled. The transformation relation is

$$\begin{Bmatrix} u_i^{'} \\ v_i^{'} \\ w_i^{'} \end{Bmatrix} = \begin{bmatrix} l_X & l_Y & l_Z \\ m_X & m_Y & m_Z \\ n_X & n_Y & n_Z \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} \tag{2.35}$$

where (l_X, l_Y, l_Z), (m_X, m_Y, m_Z) *and* (n_X, n_Y, n_Z) are the direction cosines of the local *x*, *y* and *z* coordinate axis in the global coordinate system, respectively. FEM software packages will do this transformation automatically.

Input data for bar elements:

- (*X*, *Y*, *Z*) for each node
- *E* and *A* for each element (Length *L* can be computed from the coordinates of the two nodes)

III. Beam Element

Simple Plane Beam Element

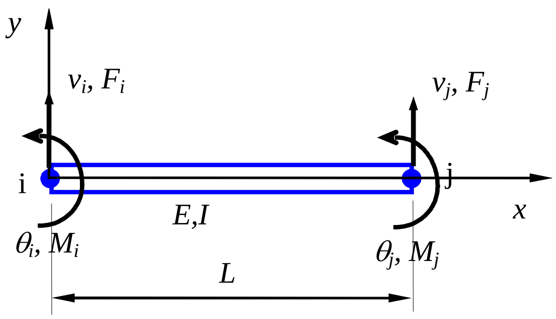


Figure 2.7. Notion for a simple beam element in 2-D.

L, I, E	length, moment of inertia of the cross-sectional area and elastic modulus
$v = v(x)$	deflection (lateral displacement) of the neutral axis of the beam
$\theta = \dfrac{dv}{dx}$	rotation of the beam about the z-axis
$Q = Q(x)$	(internal) shear force
$M = M(x)$	(internal) bending moment about z-axis
F_i, M_i, F_j, M_j	applied (external) lateral forces and moments at node i and j, respectively

Elementary Beam Theory:

We have the following results from the simple beam theory

$$EI\frac{d^2v}{dx^2} = M(x) \tag{2.36}$$

$$\frac{dM}{dx} = Q(x), \qquad \frac{dQ}{dx} = q(x) \tag{2.37}$$

where $q(x)$ is the distributed load in the lateral direction. Combining (2.36) and (2.37), we have

$$EI\frac{d^4v}{dx^4} = q(x) \tag{2.38}$$

Bending stress in the beam is given by

$$\sigma = -\frac{My}{I} \tag{2.39}$$

Simple beam theory and thus the simple beam elements are valid for "long" slender beams, for example, for beams with aspect ratios (length/height) larger than 10.

Direct Method

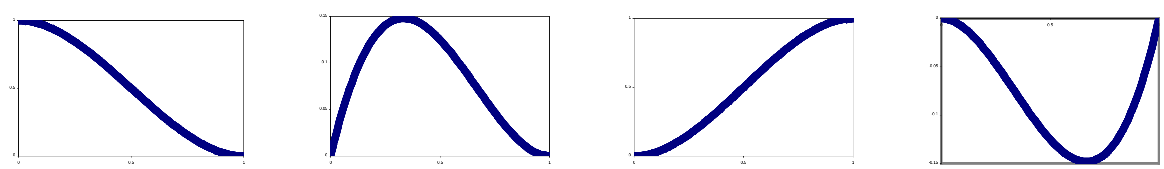
We first apply the direct method to establish the beam stiffness matrix using the results from elementary beam theory. The FE equation for a beam takes the form

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \begin{Bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{Bmatrix} = \begin{Bmatrix} F_i \\ M_i \\ F_j \\ M_j \end{Bmatrix}$$

Recall that each column in the stiffness matrix represents the forces needed to keep the structure in a special deformed shape. For example, the first column represents the forces/moments to keep the shape with $v_i = 1,\ \theta_i = v_j = \theta_j = 0$ as shown in Figure 2.8 (a). Thus, using the results from strength of materials for a cantilever beam with a force k_{11} and moment k_{21} applied at the free end, we have

$$v_i = \frac{k_{11}L^3}{3EI} - \frac{k_{21}L^2}{2EI} = 1 \qquad \text{and} \qquad \theta_i = -\frac{k_{11}L^2}{2EI} + \frac{k_{21}L}{EI} = 0$$

Solving this system of equations, we obtain k_{11} and k_{21}. Using the equilibrium conditions of the beam, we obtain k_{31} and k_{41}, and thus the first column of the stiffness matrix.



(a) $N_1(x)$ (b) $N_2(x)$ (c) $N_3(x)$ (d) $N_4(x)$

Figure 2.8. The shape of the shape function for the simple beam element.

Element stiffness equation (*local node*: i, j or 1, 2):

$$\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{Bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{Bmatrix} = \begin{Bmatrix} F_i \\ M_i \\ F_j \\ M_j \end{Bmatrix} \qquad (2.40)$$

Formal Approach

To derive the stiffness matrix in (2.40), we introduce four shape functions (Figure 2.8),

$$\begin{aligned} N_1(x) &= 1 - 3x^2/L^2 + 2x^3/L^3 \\ N_2(x) &= x - 2x^2/L + x^3/L^2 \\ N_3(x) &= 3x^2/L^2 - 2x^3/L^3 \\ N_4(x) &= -x^2/L + x^3/L^2 \end{aligned} \tag{2.41}$$

Then, we can represent the deflection as,

$$v(x) = \mathbf{Nu} = \begin{bmatrix} N_1(x) & N_2(x) & N_3(x) & N_4(x) \end{bmatrix} \begin{Bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{Bmatrix} \tag{2.42}$$

which is a cubic function. Notice that,

$$\begin{aligned} N_1 + N_3 &= 1 \\ N_2 + N_3 L + N_4 &= x \end{aligned}$$

which implies that the rigid-body motion is represented correctly by the assumed deformed shape of the beam.

Curvature of the beam is,

$$\frac{d^2 v}{dx^2} = \frac{d^2}{dx^2}\mathbf{Nu} = \mathbf{Bu} \tag{2.43}$$

where the strain-displacement matrix $\mathbf{B}$ is given by,

$$\begin{aligned} \mathbf{B} &= \frac{d^2}{dx^2}\mathbf{N} = \begin{bmatrix} N_1''(x) & N_2''(x) & N_3''(x) & N_4''(x) \end{bmatrix} \\ &= \begin{bmatrix} -\dfrac{6}{L^2} + \dfrac{12x}{L^3} & -\dfrac{4}{L} + \dfrac{6x}{L^2} & \dfrac{6}{L^2} - \dfrac{12x}{L^3} & -\dfrac{2}{L} + \dfrac{6x}{L^2} \end{bmatrix} \end{aligned} \tag{2.44}$$

Strain energy stored in the beam element is

$$\begin{aligned} U &= \frac{1}{2}\int_V \sigma^T \varepsilon dV = \frac{1}{2}\int_0^L \int_A \left(-\frac{My}{I}\right)^T \frac{1}{E}\left(-\frac{My}{I}\right) dA dx = \frac{1}{2}\int_0^L M^T \frac{1}{EI} M dx \\ &= \frac{1}{2}\int_0^L \left(\frac{d^2 v}{dx^2}\right)^T EI \left(\frac{d^2 v}{dx^2}\right) dx = \frac{1}{2}\int_0^L (\mathbf{Bu})^T EI (\mathbf{Bu}) dx \\ &= \frac{1}{2}\mathbf{u}^T \left(\int_0^L \mathbf{B}^T EI \mathbf{B} dx\right)\mathbf{u} \end{aligned}$$

We conclude that the stiffness matrix for the simple beam element is

$$\mathbf{k} = \int_0^L \mathbf{B}^T EI \mathbf{B} dx \tag{2.45}$$

Applying the result in (2.44) and carrying out the integration, we arrive at the same stiffness matrix as given in (2.40).

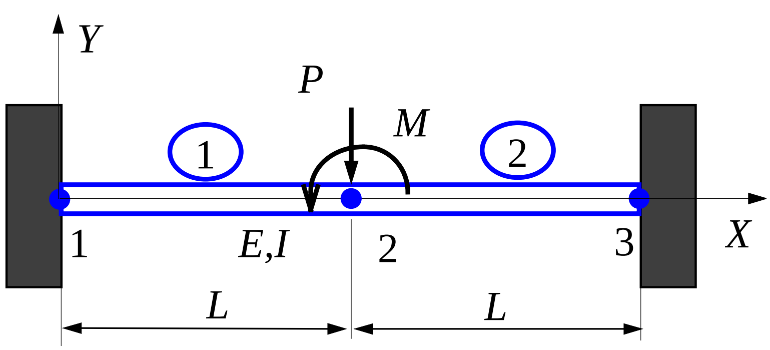
Combining the axial stiffness (from the bar element), we obtain the stiffness matrix of a *general 2-D beam element*,

$$\mathbf{k} = \begin{array}{c} \begin{matrix} \quad u_i & \quad v_i & \quad \theta_i & \quad u_j & \quad v_j & \quad \theta_j \end{matrix} \\ \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \end{array} \qquad (2.46)$$

3-D Beam Element

The element stiffness matrix is formed in the local (2-D) coordinate system first and then transformed into the global (3-D) coordinate system to be assembled. Details for 3-D beam elements can be found in the references listed at the end of the lecture notes.

Example 2.5



Given: The beam shown above is clamped at the two ends and acted upon by the force *P* and moment *M* in the mid-span.

Find: The deflection and rotation at the center node and the reaction forces and moments at the two ends.

Solution: Element stiffness matrices are

$$\mathbf{k}_1 = \frac{EI}{L^3}\begin{array}{c} \begin{array}{cccc} v_1 & \theta_1 & v_2 & \theta_2 \end{array} \\ \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \end{array}$$

$$\mathbf{k}_2 = \frac{EI}{L^3}\begin{array}{c} \begin{array}{cccc} v_2 & \theta_2 & v_3 & \theta_3 \end{array} \\ \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \end{array}$$

Global FE equation is

$$\frac{EI}{L^3}\begin{array}{c} \begin{array}{cccccc} v_1 & \theta_1 & v_2 & \theta_2 & v_3 & \theta_3 \end{array} \\ \left[\begin{array}{cc|cc|cc} 12 & 6L & -12 & 6L & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 \\ \hline -12 & -6L & 24 & 0 & -12 & 6L \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 \\ \hline 0 & 0 & -12 & -6L & 12 & -6L \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 \end{array}\right] \end{array} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} F_{1Y} \\ M_1 \\ F_{2Y} \\ M_2 \\ F_{3Y} \\ M_3 \end{Bmatrix}$$

Loads and constraints (BC's) are

$$F_{2Y} = -P, \qquad M_2 = M, \qquad v_1 = v_3 = \theta_1 = \theta_3 = 0$$

Reduced FE equation

$$\frac{EI}{L^3}\begin{bmatrix} 24 & 0 \\ 0 & 8L^2 \end{bmatrix}\begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} -P \\ M \end{Bmatrix}$$

Solving this, we obtain

$$\begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \frac{L}{24EI}\begin{Bmatrix} -PL^2 \\ 3M \end{Bmatrix}$$

From the global FE equation, we obtain the reaction forces and moments

$$\begin{Bmatrix} F_{1Y} \\ M_1 \\ F_{3Y} \\ M_3 \end{Bmatrix} = \frac{EI}{L^3}\begin{bmatrix} -12 & 6L \\ -6L & 2L^2 \\ -12 & -6L \\ 6L & 2L^2 \end{bmatrix}\begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \frac{1}{4}\begin{Bmatrix} 2P + 3M/L \\ PL + M \\ 2P - 3M/L \\ -PL + M \end{Bmatrix}$$

Stresses in the beam at the two ends can be calculated using the formula

$$\sigma = \sigma_x = -\frac{My}{I}$$

Note that the FE solution is exact for this problem according to the simple beam theory, since no distributed load is present between the nodes. Recall that (Eq. (2.38))

$$EI\frac{d^4 v}{dx^4} = q(x)$$

If $q(x)$=0, then exact solution for the deflection v is a cubic function of x, which is exactly what described by the shape functions given in (2.42).

Equivalent Nodal Loads of Distributed Transverse Load

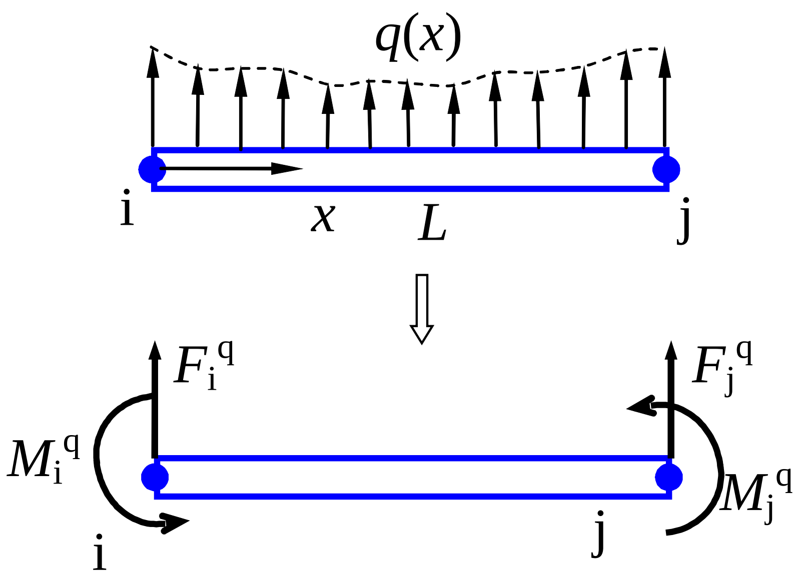


Figure 2.9. Conversion of the distributed lateral load to nodal forces and moments.

To convert a distributed load to nodal forces and moments, we consider again the work done by the distributed load q

$$W_q = \frac{1}{2}\int_0^L v(x)q(x)dx = \frac{1}{2}\int_0^L (\mathbf{Nu})^T q(x)dx = \frac{1}{2}\mathbf{u}^T\int_0^L \mathbf{N}^T q(x)dx$$

The work done by the equivalent nodal forces (and moments) is

$$W_{f_q} = \frac{1}{2}\begin{bmatrix} v_i & \theta_i & v_j & \theta_j \end{bmatrix}\begin{Bmatrix} F_i^q \\ M_i^q \\ F_j^q \\ M_j^q \end{Bmatrix} = \frac{1}{2}\mathbf{u}^T\mathbf{f}_q$$

By equating $W_q = W_{f_q}$, we obtain the equivalent nodal force vector as

$$\mathbf{f}_q = \int_0^L \mathbf{N}^T q(x) dx \tag{2.47}$$

which is valid for arbitrary distributions of $q(x)$. For constant q, we have the results shown in Figure 2.10 (verify this). An example of this result is given in Figure 2.11.

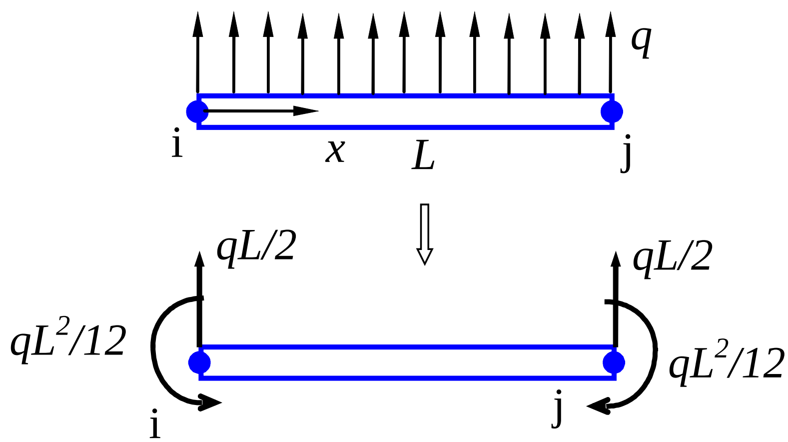


Figure 2.10. Conversion of a constant distributed lateral load to nodal forces and moments.

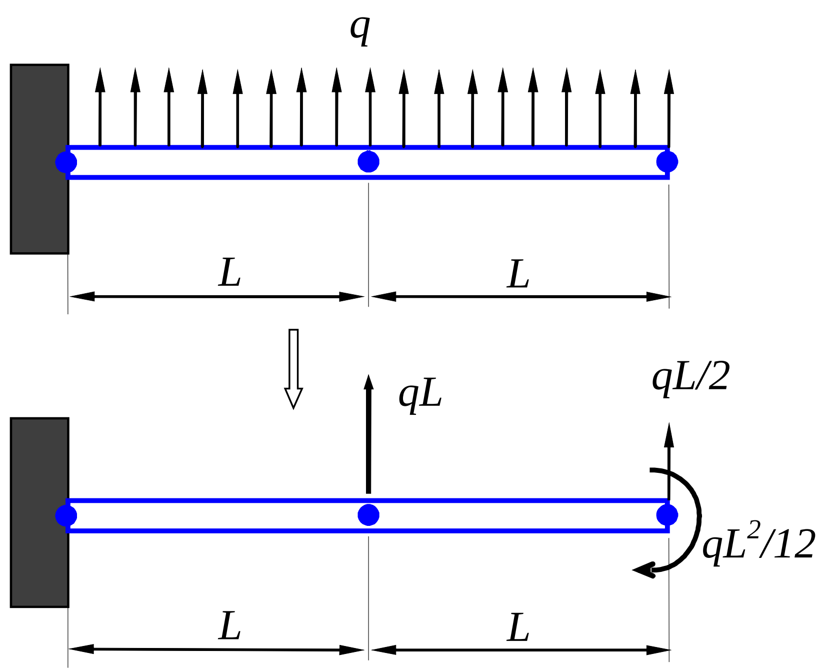
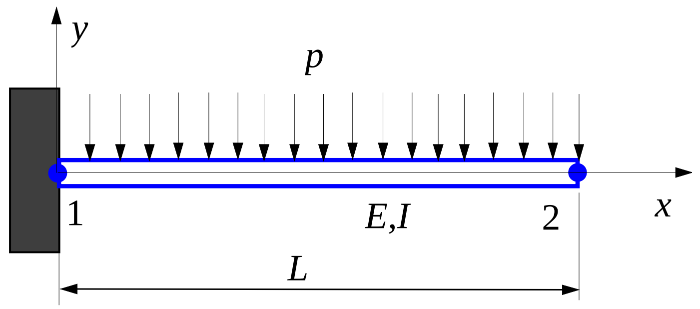


Figure 2.11. Conversion of a constant distributed lateral load on two beam elements.

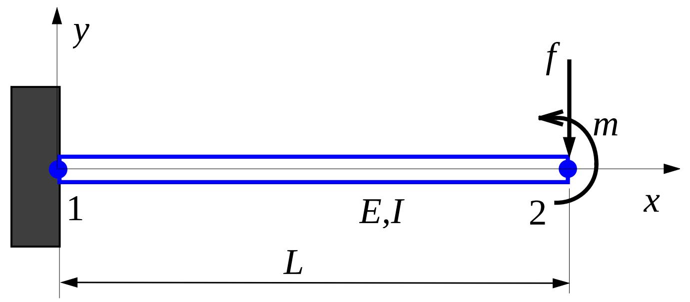
Example 2.6



Given: A cantilever beam with distributed lateral load p as shown above.

Find: The deflection and rotation at the right end, the reaction force and moment at the left end.

Solution: The work-equivalent nodal loads are shown below,



where

$$f = pL/2, \qquad m = pL^2/12$$

Applying the FE equation, we have

$$\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} F_{1Y} \\ M_1 \\ F_{2Y} \\ M_2 \end{Bmatrix}$$

Load and constraints (BCs) are

$$F_{2Y} = -f, \qquad M_2 = m$$

$$v_1 = \theta_1 = 0$$

Reduced equation is

$$\frac{EI}{L^3}\begin{bmatrix} 12 & -6L \\ -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} -f \\ m \end{Bmatrix}$$

Solving this, we obtain

$$\begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \frac{L}{6EI} \begin{Bmatrix} -2L^2 f + 3Lm \\ -3Lf + 6m \end{Bmatrix} = \begin{Bmatrix} -pL^4/8EI \\ -pL^3/6EI \end{Bmatrix} \tag{A}$$

These nodal values are the same as the exact solution. Note that the deflection $v(x)$ (for $0 < x < L$) in the beam by the FEM is, however, different from that by the exact solution. The exact solution by the simple beam theory is a 4th order polynomial of x, while the FE solution of v is only a 3rd order polynomial of x.

If the equivalent moment m is ignored, we have,

$$\begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \frac{L}{6EI} \begin{Bmatrix} -2L^2 f \\ -3Lf \end{Bmatrix} = \begin{Bmatrix} -pL^4/6EI \\ -pL^3/4EI \end{Bmatrix} \tag{B}$$

The errors in (B) will decrease if more elements are used. The equivalent moment m is often ignored in the FEM applications. The FE solutions still converge as more elements are applied.

From the FE equation, we can calculate the reaction force and moment as,

$$\begin{Bmatrix} F_{1Y} \\ M_1 \end{Bmatrix} = \frac{EI}{L^3} \begin{bmatrix} -12 & 6L \\ -6L & 2L^2 \end{bmatrix} \begin{Bmatrix} v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} pL/2 \\ 5pL^2/12 \end{Bmatrix}$$

where the result in (A) has been used. This force vector gives the total *effective nodal forces* which include the equivalent nodal forces for the distributed lateral load p given by,

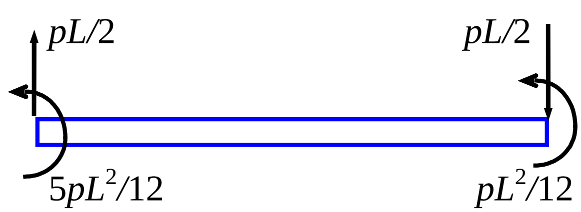
$$\begin{Bmatrix} -pL/2 \\ -pL^2/12 \end{Bmatrix}$$

The correct *reaction forces* can be obtained as follows,

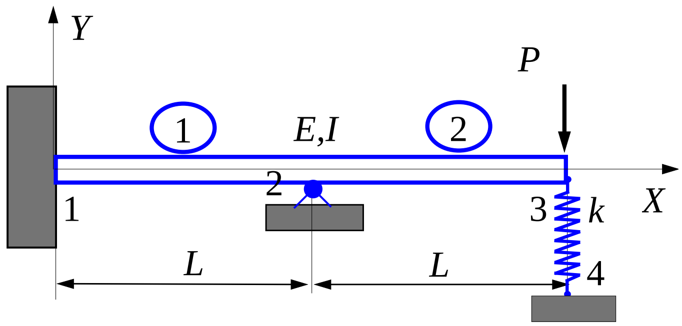
$$\begin{Bmatrix} F_{1Y} \\ M_1 \end{Bmatrix} = \begin{Bmatrix} pL/2 \\ 5pL^2/12 \end{Bmatrix} - \begin{Bmatrix} -pL/2 \\ -pL^2/12 \end{Bmatrix} = \begin{Bmatrix} pL \\ pL^2/2 \end{Bmatrix}$$

***Check the results*:**

Draw the FBD for the FE model (with the equivalent nodal force vector) and the check the equilibrium condition.



Example 2.7



Given: P = 50 kN, k = 200 kN/m, L = 3 m, E = 210 GPa, $I = 2\times10^{-4}$ m^4.

Find: Deflections, rotations and reaction forces.

Solution:

The beam has a roller (or hinge) support at node 2 and a spring support at node 3. We use two beam elements and one spring element to solve this problem.

The spring stiffness matrix is given by

$$\mathbf{k}_s = \begin{array}{c} \begin{matrix} v_3 & \quad v_4 \end{matrix} \\ \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \end{array}$$

Adding this stiffness matrix to the global FE equation (see *Example 2.5*), we have

$$\frac{EI}{L^3}\begin{array}{c} \begin{matrix} v_1 & \theta_1 & v_2 & \theta_2 & v_3 & \theta_3 & v_4 \end{matrix} \\ \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 & 0 \\ & 4L^2 & -6L & 2L^2 & 0 & 0 & 0 \\ & & 24 & 0 & -12 & 6L & 0 \\ & & & 8L^2 & -6L & 2L^2 & 0 \\ & & & & 12+k' & -6L & -k' \\ & & & & & 4L^2 & 0 \\ & \mathit{Symmetry} & & & & & k' \end{bmatrix} \end{array} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} F_{1Y} \\ M_1 \\ F_{2Y} \\ M_2 \\ F_{3Y} \\ M_3 \\ F_{4Y} \end{Bmatrix}$$

in which

$$k' = \frac{L^3}{EI}k$$

is used to simplify the notation.

We now apply the boundary conditions

$$v_1 = \theta_1 = v_2 = v_4 = 0,$$
$$M_2 = M_3 = 0, \qquad F_{3Y} = -P$$

'Deleting' the first three and seventh equations (rows and columns), we have the following reduced equation

$$\frac{EI}{L^3}\begin{bmatrix} 8L^2 & -6L & 2L^2 \\ -6L & 12+k' & -6L \\ 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} \theta_2 \\ v_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -P \\ 0 \end{Bmatrix}$$

Solving this equation, we obtain the deflection and rotations at node 2 and node 3,

$$\begin{Bmatrix} \theta_2 \\ v_3 \\ \theta_3 \end{Bmatrix} = -\frac{PL^2}{EI(12+7k')}\begin{Bmatrix} 3 \\ 7L \\ 9 \end{Bmatrix}$$

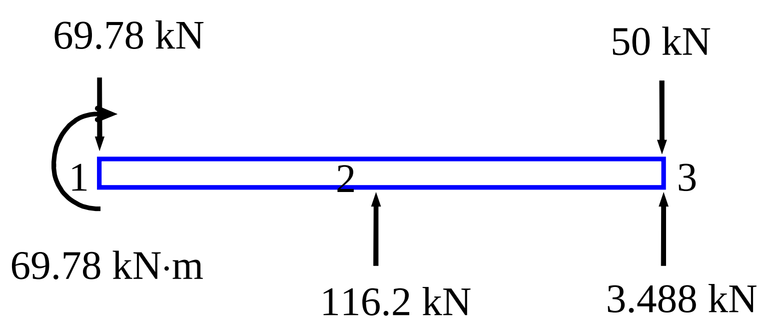
The influence of the spring k is easily seen from this result. Plugging in the given numbers, we can calculate

$$\begin{Bmatrix} \theta_2 \\ v_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -0.002492\,\text{rad} \\ -0.01744\,\text{m} \\ -0.007475\,\text{rad} \end{Bmatrix}$$

From the global FE equation, we obtain the nodal reaction forces as,

$$\begin{Bmatrix} F_{1Y} \\ M_1 \\ F_{2Y} \\ F_{4Y} \end{Bmatrix} = \begin{Bmatrix} -69.78\,\text{kN} \\ -69.78\,\text{kN}\cdot\text{m} \\ 116.2\,\text{kN} \\ 3.488\,\text{kN} \end{Bmatrix}$$

Checking the results: Draw *free body diagram* of the beam

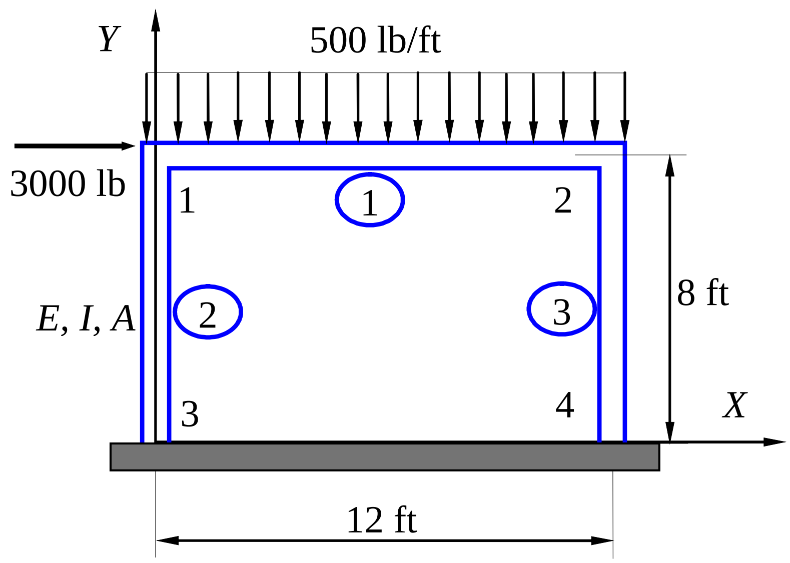


Sum the forces and moments to verify that equilibrium of the beam is satisfied.

FE Analysis of Frame Structures

Members in a frame are considered to be rigidly connected (for example, welded together). Both forces and moments can be transmitted through their joints. We need the *general beam element* (combinations of bar and simple beam elements) to model frames.

Example 2.8

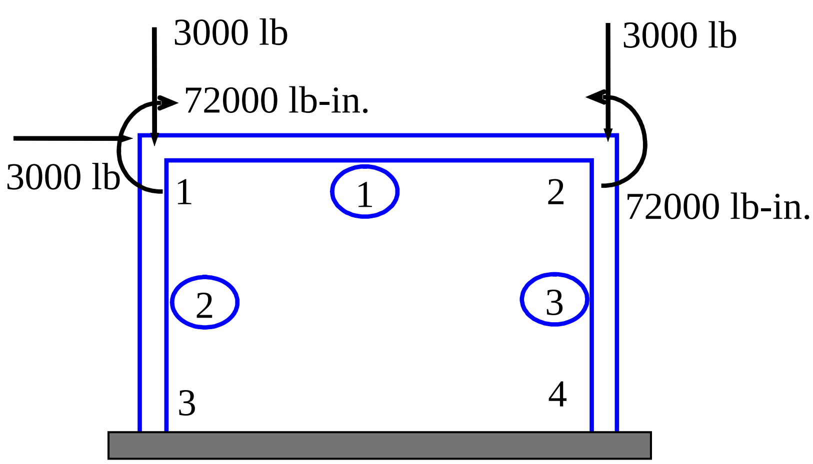


Given: $E = 30 \times 10^6$ psi, $I = 65\,\text{in.}^4$, $A = 6.8\,\text{in.}^2$

Find: Displacements and rotations of the two joints 1 and 2.

Solution:

For this example, we first convert the distributed load to its equivalent nodal loads to obtain the following FE mode.



In *local coordinate system*, the stiffness matrix for a general 2-D beam element is

$$
\begin{array}{c}
\begin{array}{cccccc} u_i & v_i & \theta_i & u_j & v_j & \theta_j \end{array} \\
\mathbf{k} = \begin{bmatrix}
\frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\
0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\
0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\
-\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\
0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\
0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L}
\end{bmatrix}
\end{array}
$$

Element Connectivity Table

Element	*Node i* (1)	*Node j* (2)
1	1	2
2	3	1
3	4	2

For element 1, we have

$$
\begin{array}{c}
\begin{array}{cccccc} u_1 & v_1 & \theta_1 & u_2 & v_2 & \theta_2 \end{array} \\
\mathbf{k}_1 = \mathbf{k}_1' = 10^4 \times \begin{bmatrix}
141.7 & 0 & 0 & -141.7 & 0 & 0 \\
0 & 0.784 & 56.4 & 0 & -0.784 & 56.4 \\
0 & 56.4 & 5417 & 0 & -56.4 & 2708 \\
-141.7 & 0 & 0 & 141.7 & 0 & 0 \\
0 & -0.784 & -56.4 & 0 & 0.784 & -56.4 \\
0 & 56.4 & 2708 & 0 & -56.4 & 5417
\end{bmatrix}
\end{array}
$$

For elements 2 and 3, the stiffness matrix in *local system* is

$$
\begin{array}{c}
\begin{array}{cccccc} u_i' & v_i' & \theta_i' & u_j' & v_j' & \theta_j' \end{array} \\
\mathbf{k}_2' = \mathbf{k}_3' = 10^4 \times \begin{bmatrix}
212.5 & 0 & 0 & -212.5 & 0 & 0 \\
0 & 2.65 & 127 & 0 & -2.65 & 127 \\
0 & 127 & 8125 & 0 & -127 & 4063 \\
-212.5 & 0 & 0 & 212.5 & 0 & 0 \\
0 & -2.65 & -127 & 0 & 2.65 & -127 \\
0 & 127 & 4063 & 0 & -127 & 8125
\end{bmatrix}
\end{array}
$$

where i = 3, j = 1 for element 2, and I = 4, j = 2 for element 3.

The transformation matrix **T** is

$$\mathbf{T} = \begin{bmatrix} l & m & 0 & 0 & 0 & 0 \\ -m & l & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & 0 \\ 0 & 0 & 0 & -m & l & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

We have $l = 0$, $m = 1$ for both elements 2 and 3. Thus,

$$\mathbf{T} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Using the transformation relation

$$\mathbf{k} = \mathbf{T}^T \mathbf{k}' \mathbf{T}$$

we obtain the stiffness matrices in the *global coordinate system* for elements 2 and 3

$$\mathbf{k}_2 = 10^4 \times \begin{array}{c} \begin{array}{cccccc} u_3 & v_3 & \theta_3 & u_1 & v_1 & \theta_1 \end{array} \\ \begin{bmatrix} 2.65 & 0 & -127 & -2.65 & 0 & -127 \\ 0 & 212.5 & 0 & 0 & -212.5 & 0 \\ -127 & 0 & 8125 & 127 & 0 & 4063 \\ -2.65 & 0 & 127 & 2.65 & 0 & 127 \\ 0 & -212.5 & 0 & 0 & 212.5 & 0 \\ -127 & 0 & 4063 & 127 & 0 & 8125 \end{bmatrix} \end{array}$$

and

$$\mathbf{k}_3 = 10^4 \times \begin{array}{c} \begin{array}{cccccc} u_4 & v_4 & \theta_4 & u_2 & v_2 & \theta_2 \end{array} \\ \begin{bmatrix} 2.65 & 0 & -127 & -2.65 & 0 & -127 \\ 0 & 212.5 & 0 & 0 & -212.5 & 0 \\ -127 & 0 & 8125 & 127 & 0 & 4063 \\ -2.65 & 0 & 127 & 2.65 & 0 & 127 \\ 0 & -212.5 & 0 & 0 & 212.5 & 0 \\ -127 & 0 & 4063 & 127 & 0 & 8125 \end{bmatrix} \end{array}$$

Assembling the global FE equation and noticing the following boundary conditions

$$u_3 = v_3 = \theta_3 = u_4 = v_4 = \theta_4 = 0$$
$$F_{1X} = 3000\,\text{lb}, \quad F_{2X} = 0, \quad F_{1Y} = F_{2Y} = -3000\,\text{lb},$$
$$M_1 = -72000\,\text{lb}\cdot\text{in.}, \quad M_2 = 72000\,\text{lb}\cdot\text{in.}$$

we obtain the condensed FE equation

$$10^4 \times \begin{bmatrix} 144.3 & 0 & 127 & -141.7 & 0 & 0 \\ 0 & 213.3 & 56.4 & 0 & -0.784 & 56.4 \\ 127 & 56.4 & 13542 & 0 & -56.4 & 2708 \\ -141.7 & 0 & 0 & 144.3 & 0 & 127 \\ 0 & -0.784 & -56.4 & 0 & 213.3 & -56.4 \\ 0 & 56.4 & 2708 & 127 & -56.4 & 13542 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} 3000 \\ -3000 \\ -72000 \\ 0 \\ -3000 \\ 72000 \end{Bmatrix}$$

Solving this, we obtain

$$\begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} 0.092\text{in.} \\ -0.00104\text{in.} \\ -0.00139\text{rad} \\ 0.0901\text{in.} \\ -0.0018\text{in.} \\ -3.88\times 10^{-5}\text{rad} \end{Bmatrix}$$

To calculate the reaction forces and moments at the two ends, we employ the element FE equations for element 2 and element 3 with known nodal displacement vectors. We obtain

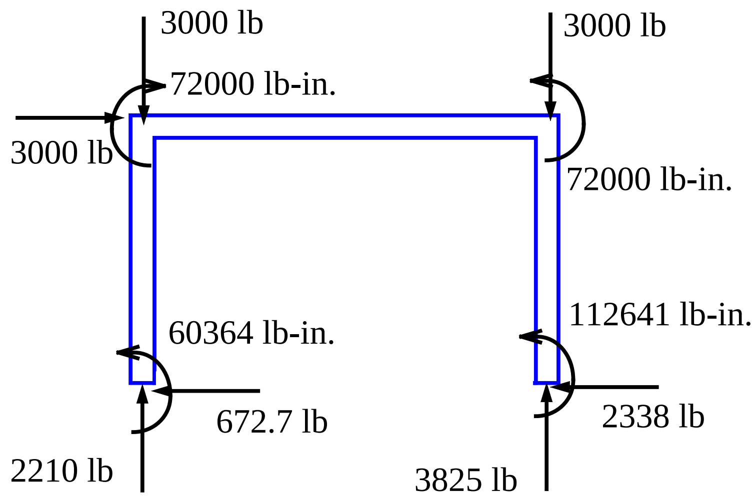
$$\begin{Bmatrix} F_{3X} \\ F_{3Y} \\ M_3 \end{Bmatrix} = \begin{Bmatrix} -672.7\text{lb} \\ 2210\text{lb} \\ 60364\,\text{lb}\cdot\text{in.} \end{Bmatrix}$$

and

$$\begin{Bmatrix} F_{4X} \\ F_{4Y} \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -2338\text{lb} \\ 3825\text{lb} \\ 112641\text{lb}\cdot\text{in.} \end{Bmatrix}$$

Check the results:

Draw the free-body diagram of the frame as shown below. Equilibrium is maintained with the calculated forces and moments. Recall that the problem we solved is the one with the equivalent loads, not the one with the distributed load. Thus the corresponding FBD for the FE model should be applied for verifying the results.



IV. Summary

In this chapter, we studied the bar elements which can be used in truss analysis and the beam elements which are used in frame analysis. The concept of the shape functions is further explored and the derivations of the stiffness matrices using the energy approach are emphasized. Treatment of distributed loads is discussed and several examples are studied.

V. Problems

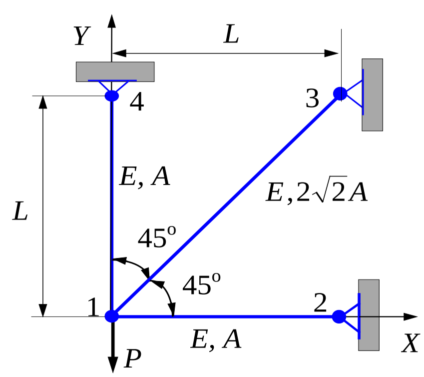
Problem 1. Using (2.47) derive the results of the equivalent nodal forces and moments for a beam element with uniformly distributed lateral load.

Problem 2. The plane truss is loaded with force P as shown below. Constants E and A for each bar are as shown in the diagram. Determine:

(a) the nodal displacement;

(b) the reaction forces;

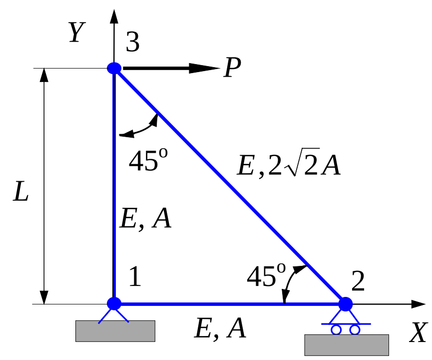
(c) the stresses in bar elements.



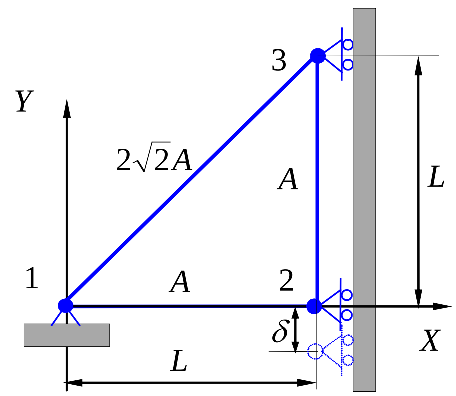
Problem 3. The plane truss is loaded with force P as shown below. Constants E and A for each bar are as shown in the diagram. Determine:

(a) the nodal displacements,

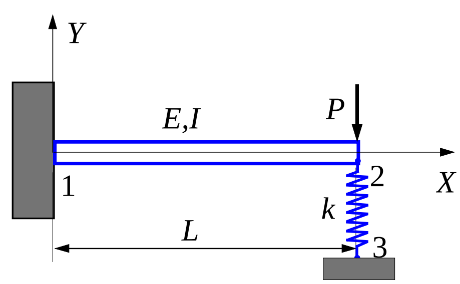
(b) the stresses in each bar elements.



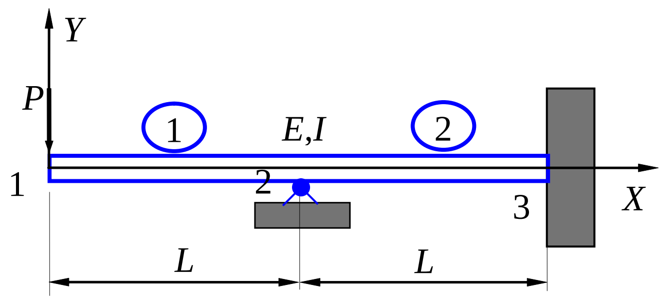
Problem 4. The plane truss is supported as shown below. The Young's modulus E is the same for all the bars. The cross-sectional areas are shown in the figure. Suppose that the node 2 settles by an amount of δ as shown. Determine the stresses in each bar element using the FEM.



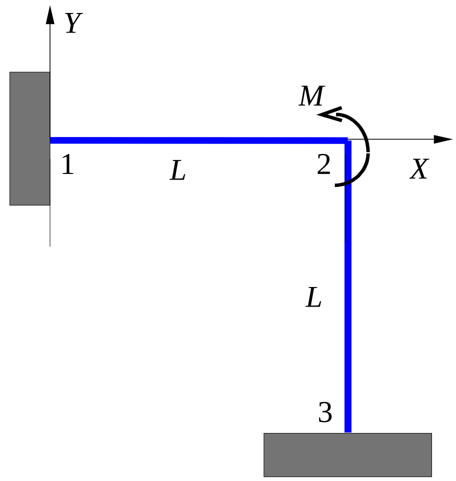
Problem 5. The cantilever beam is supported by a spring at the end as shown in the figure. Using FEM, determine the deflection and rotation at the node 2.



Problem 6. Determine the nodal displacement, rotations and reaction forces for the propped cantilever beam shown below. The beam is assumed to have constant EI and length $2L$. It is supported by a roller at midlength and is built in at the right end.



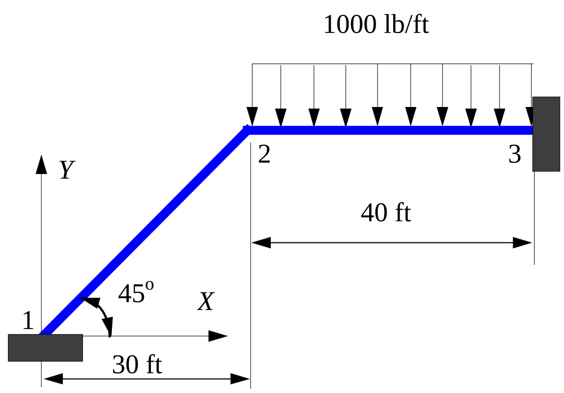
Problem 7. The 2-D frame is supported as shown in the figure. Constants *E, A, I* of the beam and the length *L* are given. Determine the displacement and rotation at node 2.



Problem 8. The plane frame is subjected to the uniformly distributed load and is fixed at the ends as shown in the figure. Assume $E = 30\times10^6$ psi, $A=$ 100 in.^{2} and I = 1000 in.^{4} for both elements of the frame. Find:

(a) the displacement and rotation of node 2;

(b) the reaction forces and moments at both ends.



Problem 9. Using an FEM software package (ANSYS, NASTRAN, or ABAQUS), solve the frame problem in Example 2.8.

Chapter 3.   Two-Dimensional Elasticity Problems

The finite element method for deformation and stress analyses of two-dimensional (2-D) structural models will be discussed in this chapter. First, the basic equations in elasticity theory (see Refs. [9, 10] for more details) for stress analysis are reviewed. Then several types of 2-D finite elements are introduced. Applications of these elements are demonstrated and their accuracies and efficiencies are discussed.

I. Stress State in Structures

In general, the stresses and strains in a structure consist of six components, that is (Figure 3.1),

$\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx}$ for stresses,

and

$\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{zx}$ for strains.

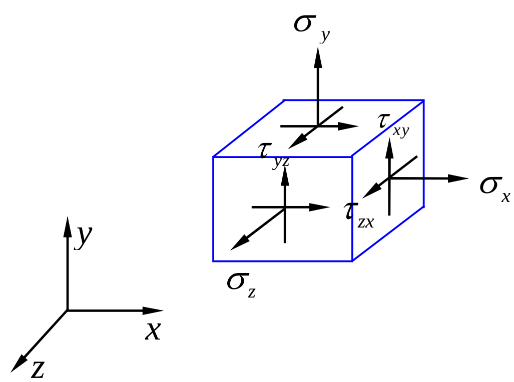


Figure 3.1. Stress components at a point in a structure.

Under certain conditions, the state of stresses and strains can be simplified. A general 3-D structure analysis can, therefore, be reduced to a 2-D analysis.

II. 2-D (Plane) Elasticity Problems

Plane stress

In the plane stress case, any *stress* component related to the *z* direction is zero, that is,

$$\sigma_z = \tau_{yz} = \tau_{zx} = 0 \qquad (\varepsilon_z \neq 0) \tag{3.1}$$

A thin planar structure with constant thickness and loading within the plane of the structure (*xy*-plane) can be regarded as a plane stress case (Figure 3.2).

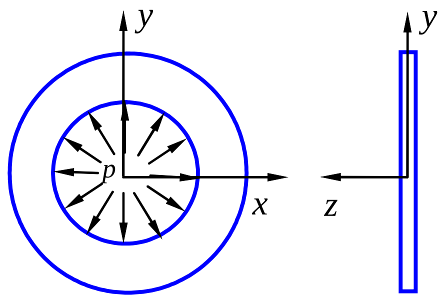


Figure 3.2. An example of a plane stress case.

Plane strain

In the plane strain case, any strain component related to the *z* direction is zero, that is,

$$\varepsilon_z = \gamma_{yz} = \gamma_{zx} = 0 \qquad\qquad (\sigma_z \neq 0) \tag{3.2}$$

A long structure with a uniform cross section and transverse loading along its length (*z*-direction), such as a tunnel, can be regarded as a plane strain case (Figure 3.3).

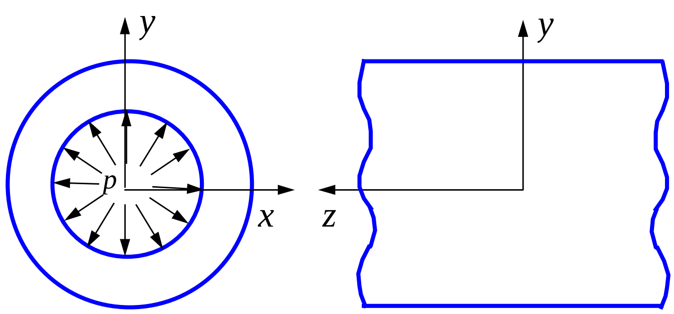


Figure 3.3. An example of a plane strain case.

Stress-Strain-Temperature (Constitutive) Relations

For elastic and isotropic materials, we have the following stress-strain relation for 2-D cases

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} 1/E & -\nu/E & 0 \\ -\nu/E & 1/E & 0 \\ 0 & 0 & 1/G \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} + \begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} \tag{3.3}$$

or,

$$\varepsilon = \mathbf{E}^{-1}\sigma + \varepsilon_0$$

where ε_0 is the initial strain (for example, due to a temperature change), E the Young's modulus, ν the Poisson's ratio and G the shear modulus. Note that

$$G = \frac{E}{2(1+\nu)} \tag{3.4}$$

which means that there are only two independent materials constants for *homogeneous* and *isotropic* materials.

We can also express stresses in terms of strains by solving the above equation

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} \right) \tag{3.5}$$

or,

$$\sigma = \mathbf{E}\varepsilon + \sigma_0$$

where $\sigma_0 = -\mathbf{E}\varepsilon_0$ is the initial stress.

The above relations are valid for *plane stress* case. For *plane strain* case, we need to replace the material constants in the above equations in the following fashion

$$E \to \frac{E}{1-\nu^2}; \qquad \nu \to \frac{\nu}{1-\nu}; \qquad G \to G \tag{3.6}$$

For example, the stress is related to strain by

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} \right)$$

in the *plane strain* case.

Initial strain due to a *temperature change* (thermal loading) is given by the following for the plane stress case

$$\begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} = \begin{Bmatrix} \alpha\Delta T \\ \alpha\Delta T \\ 0 \end{Bmatrix} \tag{3.7}$$

where α is the coefficient of thermal expansion, ΔT the change of temperature. For the plane strain case, α should be replaced by $(1+\nu)\alpha$ in (3.7). Note that if the structure is free to deform under thermal loading, there will be no (elastic) stresses in the structure.

Strain and Displacement Relations

For small strains and small rotations, we have,

$$\varepsilon_x = \frac{\partial u}{\partial x}, \qquad \varepsilon_y = \frac{\partial v}{\partial y}, \qquad \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

In matrix form, we write

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \partial/\partial x & 0 \\ 0 & \partial/\partial y \\ \partial/\partial y & \partial/\partial x \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}, \qquad \text{or} \qquad \varepsilon = \mathbf{Du} \tag{3.8}$$

From this relation, we know that the strains (and thus stresses) are one order lower than the displacements, if the displacements are represented by polynomials.

Equilibrium Equations

In elasticity theory, the stresses in the structure must satisfy the following equilibrium equations

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + f_x &= 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + f_y &= 0 \end{aligned} \tag{3.9}$$

where f_x and f_y are body forces per unit volume (such as gravity forces). In the FEM, these equilibrium conditions are satisfied in an approximate sense.

Boundary Conditions

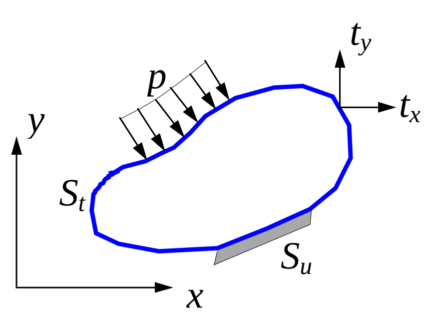


Figure 3.4. Boundary conditions for a structure.

The boundary S of the body can be divided into two parts, S_u and S_t (Figure 3.4). The boundary conditions (BCs) can be described as

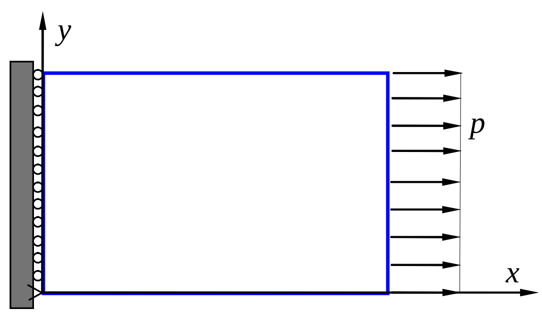
$$\begin{aligned} &u = \bar{u}, \; v = \bar{v}, && \text{on } S_u \\ &t_x = \bar{t}_x, \; t_y = \bar{t}_y, && \text{on } S_t \end{aligned} \tag{3.10}$$

in which t_x and t_y are tractions (stresses on the boundary) and the barred quantities are those with known values. In the FEM, all types of loads (distributed surface loads, body forces, concentrated forces and moments, etc.) are converted to point forces acting at the nodes.

Exact Elasticity Solution

The exact solution (displacements, strains and stresses) of a given problem must satisfy the equilibrium equations (9), the given boundary conditions (10) and compatibility conditions (structures should deform in a continuous manner, no cracks or overlaps in the obtained displacement fields).

Example 3.1



A plate is supported and loaded with distributed force p as shown in the figure. The material constants are E and ν.

The exact solution for this simple problem can be found easily as follows.

Displacement: $u = \dfrac{p}{E} x, \qquad v = -\nu \dfrac{p}{E} y$

Strain: $\varepsilon_x = \dfrac{p}{E}, \qquad \varepsilon_y = -\nu \dfrac{p}{E}, \qquad \gamma_{xy} = 0$

Stress: $\sigma_x = p, \qquad \sigma_y = 0, \qquad \tau_{xy} = 0$

Exact (or analytical) solutions for *simple* problems are numbered (suppose there is a hole in the plate or the roller support are replaced by clamped ones!). That is why we need the FEM!

III. Finite Elements for 2-D Problems

A General Formula for the Stiffness Matrix

Displacements (u, v) in a plane element are interpolated from nodal displacements (u_i, v_i) using shape functions N_i as follows,

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \cdots \\ 0 & N_1 & 0 & N_2 & \cdots \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \vdots \end{Bmatrix} \quad \text{or} \qquad \mathbf{u} = \mathbf{N}\mathbf{d} \tag{3.11}$$

where **N** is the *shape function matrix*, **u** the displacement vector and **d** the *nodal* displacement vector. Here we have assumed that u depends on the nodal values of u only, and v on nodal values of v only.

From strain-displacement relation (Eq.(3.8)), the strain vector is,

$$\varepsilon = \mathbf{D}\mathbf{u} = \mathbf{D}\mathbf{N}\mathbf{d}, \qquad \text{or} \qquad \varepsilon = \mathbf{B}\mathbf{d} \tag{3.12}$$

where $\mathbf{B} = \mathbf{DN}$ is the *strain-displacement matrix*.

Consider the strain energy stored in an element,

$$\begin{aligned} U &= \frac{1}{2}\int_V \sigma^T \varepsilon dV = \frac{1}{2}\int_V \left(\sigma_x \varepsilon_x + \sigma_y \varepsilon_y + \tau_{xy}\gamma_{xy}\right) dV \\ &= \frac{1}{2}\int_V (\mathbf{E}\varepsilon)^T \varepsilon dV = \frac{1}{2}\int_V \varepsilon^T \mathbf{E}\varepsilon dV = \frac{1}{2}\mathbf{d}^T \int_V \mathbf{B}^T\mathbf{E}\mathbf{B} dV \mathbf{d} \\ &= \frac{1}{2}\mathbf{d}^T \mathbf{k}\mathbf{d} \end{aligned}$$

From this, we obtain the general formula for the *element stiffness matrix*

$$\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E}\mathbf{B} dV \tag{3.13}$$

Note that unlike the 1-D cases, **E** here is a *matrix* which is given by the stress-strain relation (e.g., Eq.(3.5) for plane stress).

The stiffness matrix **k** defined by (3.13) is symmetric since **E** is symmetric. Also note that given the material property, the behavior of **k** depends on the **B** matrix only, which in turn on the shape functions. Thus, the quality of finite elements in representing the behavior of a structure is mainly determined by the choice of shape functions. Most commonly employed 2-D elements are linear or quadratic triangles and quadrilaterals.

Constant Strain Triangle (CST or T3)

This is the simplest 2-D element (Figure 3.5), which is also called ***linear triangular element***.

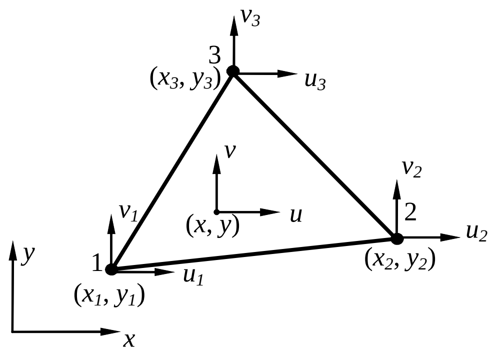


Figure 3.5. Linear triangular element (T3).

For this element, we have three nodes at the vertices of the triangle, which are numbered around the element in the counterclockwise direction. Each node has two degrees of freedom (can move in the x and y directions). The displacements u and v are assumed to be linear functions within the element, that is,

$$u = b_1 + b_2 x + b_3 y, \qquad v = b_4 + b_5 x + b_6 y \tag{3.14}$$

where b_i (i = 1, 2, ..., 6) are constants. From these, the strains are found to be,

$$\varepsilon_x = b_2, \qquad \varepsilon_y = b_6, \qquad \gamma_{xy} = b_3 + b_5 \tag{3.15}$$

which are constant throughout the element. Thus, we have the name "constant strain triangle" (CST).

Displacements given by (3.14) should satisfy the following six equations

$$\begin{aligned} u_1 &= b_1 + b_2 x_1 + b_3 y_1 \\ u_2 &= b_1 + b_2 x_2 + b_3 y_2 \\ &\vdots \\ v_3 &= b_4 + b_5 x_3 + b_6 y_3 \end{aligned}$$

Solving these equations, we can find the coefficients b_1, b_2, ..., and b_6 in terms of nodal displacements and coordinates. Substituting these coefficients into (3.14) and rearranging the terms, we obtain

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \tag{3.16}$$

where the shape functions (linear functions in x and y) are

$$\begin{aligned} N_1 &= \frac{1}{2A}\{(x_2 y_3 - x_3 y_2) + (y_2 - y_3)x + (x_3 - x_2)y\} \\ N_2 &= \frac{1}{2A}\{(x_3 y_1 - x_1 y_3) + (y_3 - y_1)x + (x_1 - x_3)y\} \\ N_3 &= \frac{1}{2A}\{(x_1 y_2 - x_2 y_1) + (y_1 - y_2)x + (x_2 - x_1)y\} \end{aligned} \tag{3.17}$$

and

$$A = \frac{1}{2}\det\begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \tag{3.18}$$

is the area of the triangle (Prove this!).

Using the strain-displacement relation (3.8), results (3.16) and (3.17), we have

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \mathbf{Bd} = \frac{1}{2A}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \tag{3.19}$$

where $x_{ij} = x_i - x_j$ and $y_{ij} = y_i - y_j$ (i, j = 1, 2, 3). Again, we see constant strains within the element. From stress-strain relation (Eq.(3.5), for example), we see that stresses obtained using the CST element are also constant.

Applying formula (3.13), we obtain the element stiffness matrix for the CST element

$$\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E} \mathbf{B} dV = tA(\mathbf{B}^T \mathbf{E} \mathbf{B}) \tag{3.20}$$

in which t is the thickness of the element. Notice that **k** for CST is a 6 by 6 *symmetric* matrix.

Both the expressions of the shape functions in (3.17) and their derivations are lengthy and offer little insight into the behavior of the element.

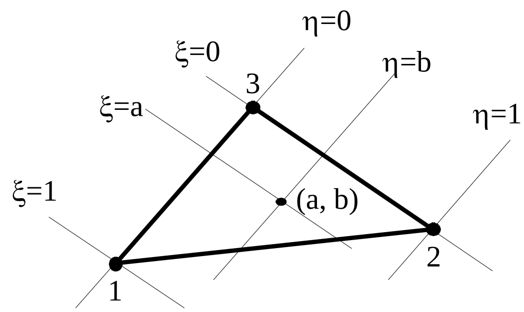


Figure 3.6. The natural coordinate system defined on the triangle.

We introduce the *natural coordinates* (ξ,η) on the triangle (Figure 3.6). Then *the shape functions* can be represented simply by

$$N_1 = \xi, N_2 = \eta, N_3 = 1 - \xi - \eta \tag{3.21}$$

Notice that,

$$N_1 + N_2 + N_3 = 1 \tag{3.22}$$

which ensures that the rigid-body translation is represented by the chosen shape functions. Also, as in the 1-D case,

$$N_i = \begin{cases} 1, & \text{at node i;} \\ 0, & \text{at the other nodes} \end{cases} \tag{3.23}$$

and varies linearly within the element. The plot for shape function N_1 is shown in Figure 3.7. N_2 and N_3 have similar features.

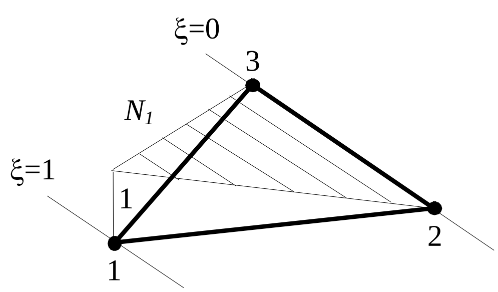


Figure 3.7. Plot of the shape function N_1 for T3 element.

We have two coordinate systems for the element: the global coordinates (x, y) and the natural (local) coordinates (ξ,η). The relation between the two is given by

$$\begin{aligned} x &= N_1 x_1 + N_2 x_2 + N_3 x_3 \\ y &= N_1 y_1 + N_2 y_2 + N_3 y_3 \end{aligned} \qquad (3.24)$$

or,

$$\begin{aligned} x &= x_{13}\xi + x_{23}\eta + x_3 \\ y &= y_{13}\xi + y_{23}\eta + y_3 \end{aligned} \qquad (3.25)$$

where $x_{ij} = x_i - x_j$ and $y_{ij} = y_i - y_j$ (i, j = 1, 2, 3) as defined earlier.

Displacement u or v on the element can be viewed as functions of (x, y) or (ξ,η). Using the chain rule for derivatives, we have,

$$\begin{Bmatrix} \dfrac{\partial u}{\partial \xi} \\ \dfrac{\partial u}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \end{Bmatrix} = \mathbf{J} \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \end{Bmatrix} \qquad (3.26)$$

where $\mathbf{J}$ is called the *Jacobian matrix* of the transformation.

From (3.25), we calculate

$$\mathbf{J} = \begin{bmatrix} x_{13} & y_{13} \\ x_{23} & y_{23} \end{bmatrix}, \qquad \mathbf{J}^{-1} = \frac{1}{2A} \begin{bmatrix} y_{23} & -y_{13} \\ -x_{23} & x_{13} \end{bmatrix} \qquad (3.27)$$

where $\det \mathbf{J} = x_{13}y_{23} - x_{23}y_{13} = 2A$ has been used (A is the area of the triangle. Prove this!).

From (3.26), (3.27), (3.16) and (3.21) we have

$$\begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} y_{23} & -y_{13} \\ -x_{23} & x_{13} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial u}{\partial \xi} \\ \dfrac{\partial u}{\partial \eta} \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} y_{23} & -y_{13} \\ -x_{23} & x_{13} \end{bmatrix} \begin{Bmatrix} u_1 - u_3 \\ u_2 - u_3 \end{Bmatrix} \qquad (3.28)$$

Similarly,

$$\begin{Bmatrix} \dfrac{\partial v}{\partial x} \\ \dfrac{\partial v}{\partial y} \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} y_{23} & -y_{13} \\ -x_{23} & x_{13} \end{bmatrix} \begin{Bmatrix} v_1 - v_3 \\ v_2 - v_3 \end{Bmatrix} \qquad (3.29)$$

Using the results in (3.28) and (3.29), and the relations $\varepsilon = \mathbf{Du} = \mathbf{DNd} = \mathbf{Bd}$, we obtain the strain-displacement matrix,

$$\mathbf{B} = \frac{1}{2A}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix} \tag{3.30}$$

which is the same as we derived earlier in (3.19).

***Applications of the CST Element*:**

- Use in areas where the strain gradient is small.
- Use in mesh transition areas (fine mesh to coarse mesh).
- Avoid using CST in stress concentration or other crucial areas in the structure, such as edges of holes and corners.
- Recommended for quick and preliminary FE analysis of 2-D problems.

Linear Strain Triangle (LST or T6)

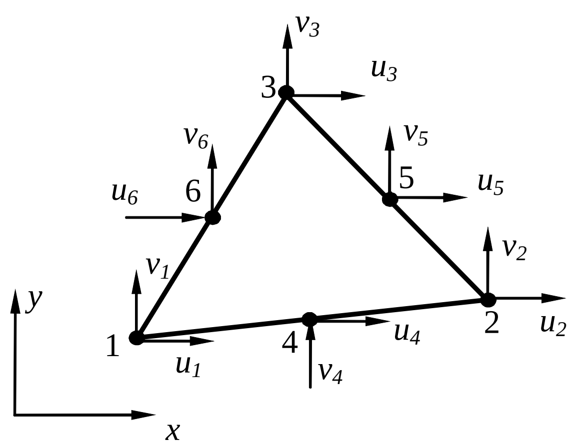


Figure 3.8. Quadratic triangular element (T6).

This type of elements (Figure 3.8) is also called *quadratic triangular element*. There are six nodes on this element: three corner nodes and three mid-side nodes. Each node has two degrees of freedom (DOFs) as before. The displacements (u, v) are assumed to be quadratic functions of (x, y),

$$\begin{aligned} u &= b_1 + b_2 x + b_3 y + b_4 x^2 + b_5 xy + b_6 y^2 \\ v &= b_7 + b_8 x + b_9 y + b_{10} x^2 + b_{11} xy + b_{12} y^2 \end{aligned} \tag{3.31}$$

where b_i (i = 1, 2, ..., 12) are constants. From these, the strains are found to be

$$
\begin{aligned}
\varepsilon_x &= b_2 + 2b_4 x + b_5 y \\
\varepsilon_y &= b_9 + b_{11} x + 2b_{12} y \\
\gamma_{xy} &= (b_3 + b_8) + (b_5 + 2b_{10})x + (2b_6 + b_{11})y
\end{aligned}
\tag{3.32}
$$

which are linear functions. Thus, we have the "linear strain triangle" (LST), which provides better results than the CST.

In the natural coordinate system we defined earlier, the six shape functions for the LST element are

$$
\begin{aligned}
N_1 &= \xi(2\xi - 1) \\
N_2 &= \eta(2\eta - 1) \\
N_3 &= \zeta(2\zeta - 1) \\
N_4 &= 4\xi\eta \\
N_5 &= 4\eta\zeta \\
N_6 &= 4\zeta\xi
\end{aligned}
\tag{3.33}
$$

in which $\zeta = 1 - \xi - \eta$. Each of these six shape functions represents a quadratic form on the element as shown in Figure 3.9.

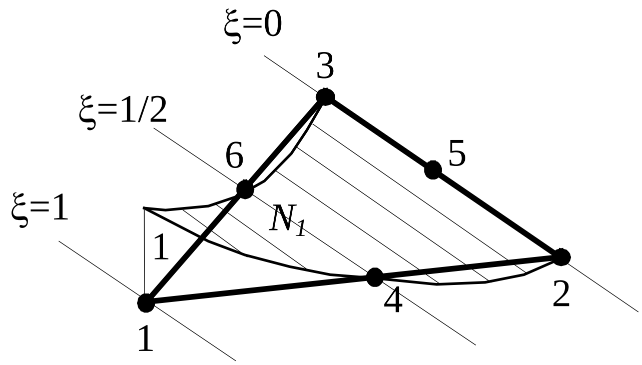


Figure 3.9. Plot of the shape function N_1 for T6 element.

Displacements can be written as,

$$
u = \sum_{i=1}^{6} N_i u_i, \qquad v = \sum_{i=1}^{6} N_i v_i \tag{3.34}
$$

The element stiffness matrix is still given by $\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E} \mathbf{B} dV$, but here $\mathbf{B}^T \mathbf{E} \mathbf{B}$ is quadratic in *x* and *y*. In general, the integral has to be computed numerically.

Linear Quadrilateral Element (Q4)

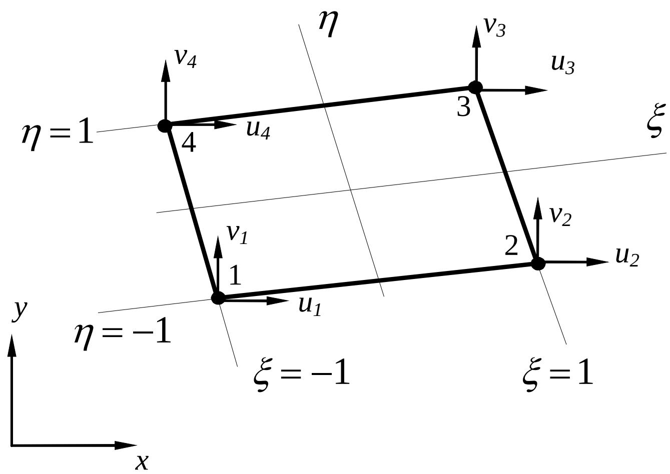


Figure 3.10. Linear quadrilateral element (Q4).

There are four nodes at the corners of the quadrilateral element (Figure 3.10). In the natural coordinate system (ξ, η), the four shape functions are,

$$\begin{aligned}
N_1 &= \frac{1}{4}(1-\xi)(1-\eta) \\
N_2 &= \frac{1}{4}(1+\xi)(1-\eta) \\
N_3 &= \frac{1}{4}(1+\xi)(1+\eta) \\
N_4 &= \frac{1}{4}(1-\xi)(1+\eta)
\end{aligned} \tag{3.35}$$

Note that $\sum_{i=1}^{4} N_i = 1$ at any point inside the element, as expected.

The displacement field is given by

$$u = \sum_{i=1}^{4} N_i u_i, \qquad v = \sum_{i=1}^{4} N_i v_i \tag{3.36}$$

which are bilinear functions over the element. The stress and strain fields are constant on this type of elements.

Quadratic Quadrilateral Element (Q8)

This is the most widely used element for 2-D problems due to its high accuracy in analysis and flexibility in modeling.

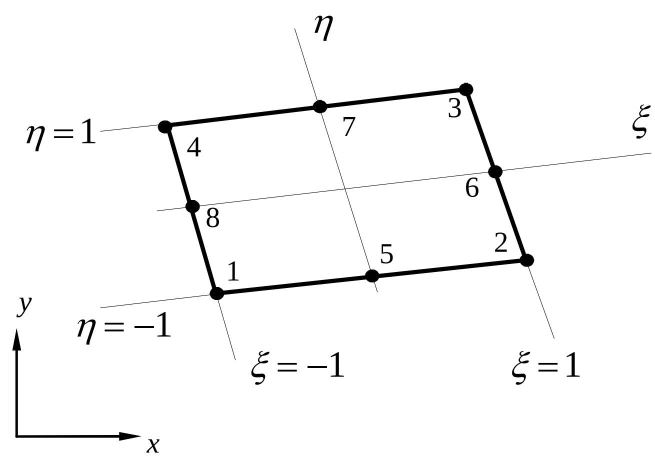


Figure 3.11. Quadratic quadrilateral element (Q8).

There are eight nodes for this element (Figure 3.11), four corners nodes and four midside nodes. In the natural coordinate system (ξ,η), the eight shape functions are,

$$
\begin{aligned}
N_1 &= \frac{1}{4}(1-\xi)(\eta-1)(\xi+\eta+1) \\
N_2 &= \frac{1}{4}(1+\xi)(\eta-1)(\eta-\xi+1) \\
N_3 &= \frac{1}{4}(1+\xi)(1+\eta)(\xi+\eta-1) \\
N_4 &= \frac{1}{4}(\xi-1)(\eta+1)(\xi-\eta+1) \\
N_5 &= \frac{1}{2}(1-\eta)(1-\xi^2) \\
N_6 &= \frac{1}{2}(1+\xi)(1-\eta^2) \\
N_7 &= \frac{1}{2}(1+\eta)(1-\xi^2) \\
N_8 &= \frac{1}{2}(1-\xi)(1-\eta^2)
\end{aligned}
\tag{3.37}
$$

Again, we have $\sum_{i=1}^{8} N_i = 1$ at any point inside the element.

The displacement field is given by

$$
u = \sum_{i=1}^{8} N_i u_i, \qquad v = \sum_{i=1}^{8} N_i v_i \tag{3.38}
$$

which are quadratic functions over the element. Strains and stresses over a quadratic quadrilateral element are linear functions, which are better representations. A model of fiber-reinforced composite materials using the Q8 elements is shown in Figure 3.12.

Notes:

- Q4 and T3 are usually used together in a mesh with linear elements.
- Q8 and T6 are usually applied in a mesh composed of quadratic elements.
- Quadratic elements are preferred for stress analysis, because of their high accuracy and the flexibility in modeling complex geometry, such as curved boundaries.

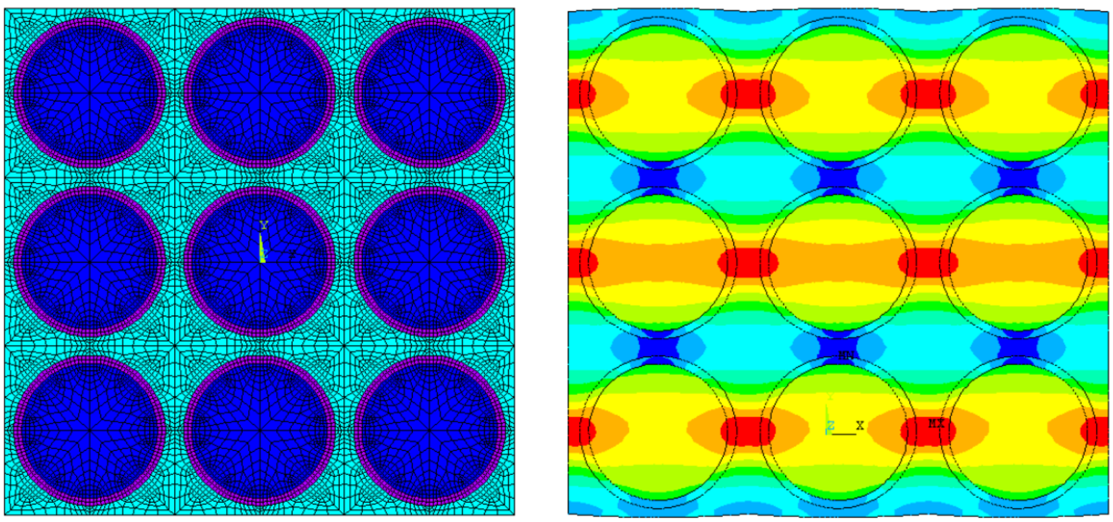


Figure 3.12. Analysis of composite materials (mesh and contour stress plots).

Transformation of Loads

Concentrated load (point forces), surface traction (pressure loads) and body force (weight) are the main types of loads applied to a structure. Both traction and body forces need to be converted to nodal forces in the FE model, since they cannot be applied to the FE model directly. The conversions of these loads are based on the same idea (the equivalent-work concept) which we have used for the cases of bar and beam elements.

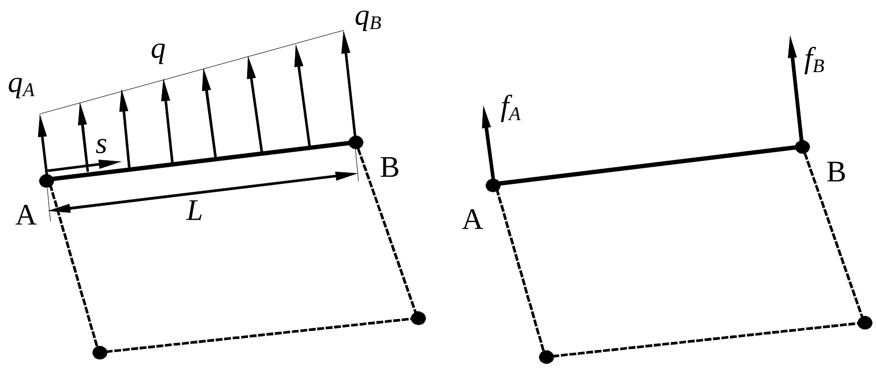


Figure 3.13. Traction applied on the edge of a Q4 element.

Suppose, for example, we have a linearly varying traction q on a Q4 element edge, as shown in the Figure 3.13. The traction is normal to the boundary. Using the local (tangential) coordinate s, we can write the work done by the traction q as

$$W_q = \frac{1}{2} t \int_0^L u_n(s) q(s) ds$$

where t is the thickness, L the side length and u_n the component of displacement normal to the edge AB.

For the Q4 element (linear displacement field), we have

$$u_n(s) = (1 - s/L)u_{nA} + (s/L)u_{nB}$$

The traction $q(s)$, which is also linear, is given in a similar way

$$q(s) = (1 - s/L)q_A + (s/L)q_B$$

Thus, we have,

$$\begin{aligned}
W_q &= \frac{1}{2} t \int_0^L \left(\begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} \begin{bmatrix} 1 - s/L \\ s/L \end{bmatrix} \right) \left(\begin{bmatrix} 1 - s/L & s/L \end{bmatrix} \begin{bmatrix} q_A \\ q_B \end{bmatrix} \right) ds \\
&= \frac{1}{2} \begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} t \int_0^L \begin{bmatrix} (1 - s/L)^2 & (s/L)(1 - s/L) \\ (s/L)(1 - s/L) & (s/L)^2 \end{bmatrix} ds \begin{bmatrix} q_A \\ q_B \end{bmatrix} \\
&= \frac{1}{2} \begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} \frac{tL}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{bmatrix} q_A \\ q_B \end{bmatrix} \\
&= \frac{1}{2} \begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} \begin{Bmatrix} f_A \\ f_B \end{Bmatrix}
\end{aligned}$$

and hence the equivalent nodal force vector is

$$\begin{Bmatrix} f_A \\ f_B \end{Bmatrix} = \frac{tL}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{Bmatrix} q_A \\ q_B \end{Bmatrix}$$

Note, for constant q, we have

$$\begin{Bmatrix} f_A \\ f_B \end{Bmatrix} = \frac{qtL}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

For quadratic elements (either triangular or quadrilateral), the traction is converted to forces at three nodes along the edge, instead of two nodes. Traction tangent to the boundary, as well as body forces, are converted to nodal forces in a similar way.

Stress Calculation

The stress in an element is determined by the following relation,

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \mathbf{E} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \mathbf{EBd} \tag{3.39}$$

where **B** is the strain-nodal displacement matrix and **d** is the nodal displacement vector which is known for each element once the global FE equation has been solved.

Stresses can be evaluated at any point inside the element (such as the center) or at the nodes. Contour plots are usually used in FEA software packages (during post-process) for users to visually inspect the stress results.

The von Mises Stress:

The von Mises stress is the *effective* or *equivalent* stress for 2-D and 3-D stress analysis. For a ductile material, the stress level is considered to be safe, if

$$\sigma_e \le \sigma_Y$$

where σ_e is the von Mises stress and σ_Y the yield stress of the material. This is a generalization of the 1-D (experimental) result to 2-D and 3-D situations.

The von Mises stress is defined by

$$\sigma_e = \frac{1}{\sqrt{2}} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2} \tag{3.40}$$

in which σ_1, σ_2 and σ_3 are the three principle stresses at the considered point in a structure.

For 2-D problems, the two principle stresses in the plane are determined by

$$
\begin{aligned}
\sigma_1^{\,P} &= \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2} \\
\sigma_2^{\,P} &= \frac{\sigma_x + \sigma_y}{2} - \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2}
\end{aligned}
\tag{3.41}
$$

Thus, we can also express the von Mises stress in terms of the stress components in the *xy* coordinate system. For plane stress conditions, we have,

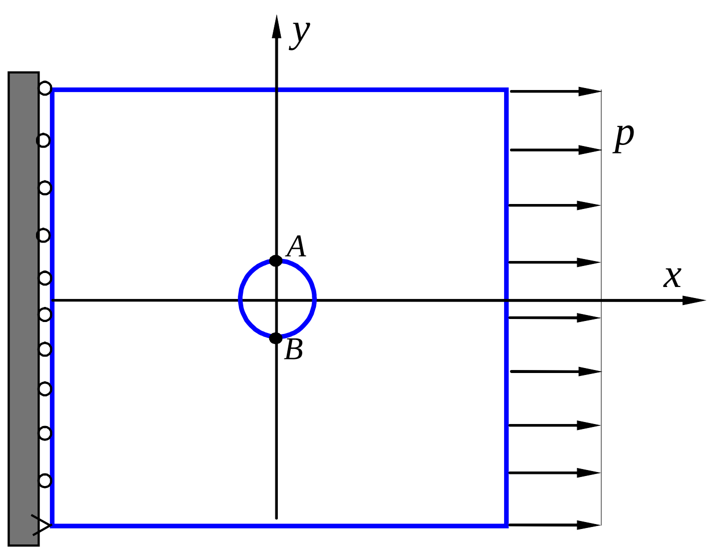
$$
\sigma_e = \sqrt{(\sigma_x + \sigma_y)^2 - 3(\sigma_x \sigma_y - \tau_{xy}^2)} \tag{3.42}
$$

Averaged Stresses:

Stresses are usually averaged at nodes in FEA software packages to provide more accurate stress values. This option should be turned off at nodes between two materials or other geometry discontinuity locations where stress discontinuity does exist.

Example 3.2

A square plate with a hole at the center is under a tension load p in x direction as shown in the figure.

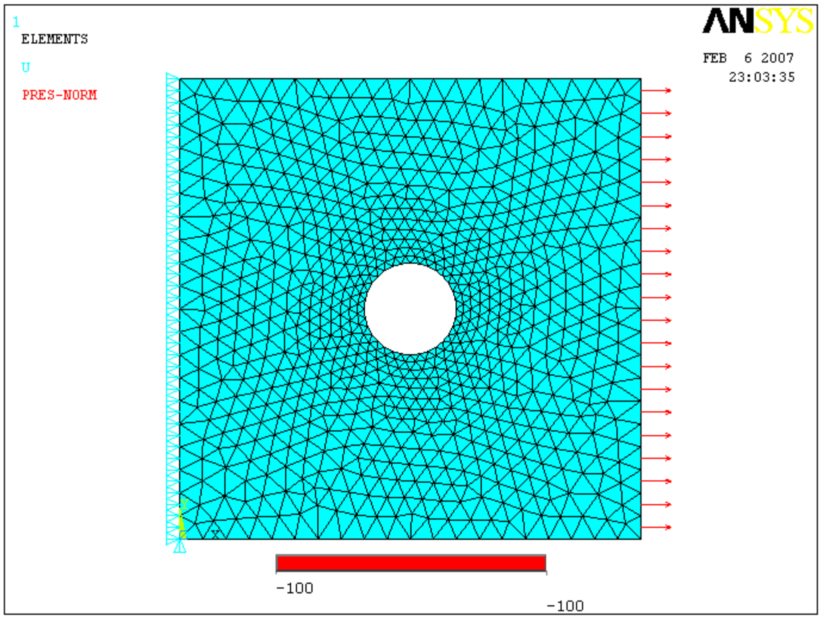


The dimension of the plate is 10 in. x 10 in., thickness is 0.1 in. and radius of the hole is 1 in. Assume $E = 10\text{x}10^6$ psi, $v = 0.3$ and $p = 100$ psi. Find the maximum stress in the plate.

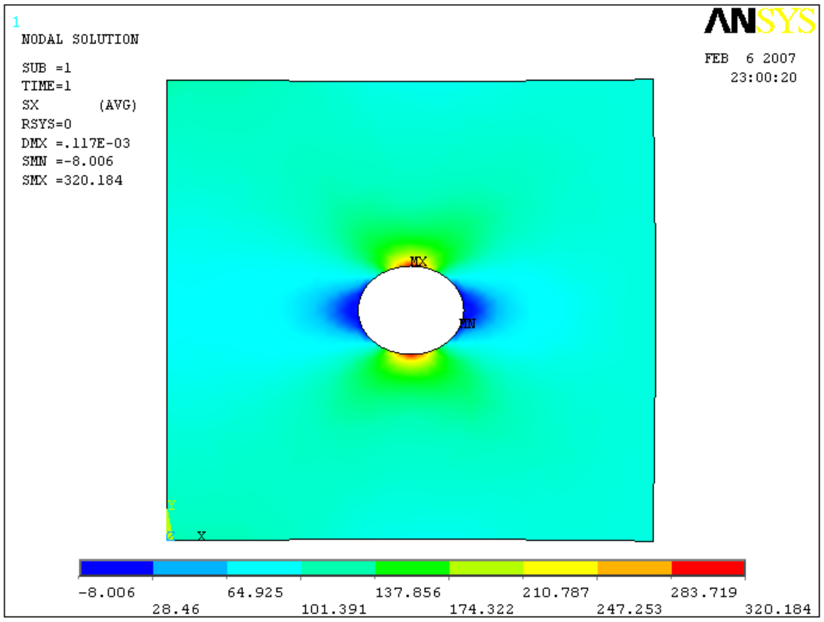
FE Analysis:

This is a plane stress case. From the knowledge of stress concentrations, we should expect the maximum stresses occur at points A and B on the edge of the hole. Value of this stress should be around 3*p* (= 300 psi) which is the exact solution for an infinitely large plate with a circular hole.

We use the *ANSYS* software to do the modeling (meshing) and analysis, using quadratic triangular (T6), linear quadrilateral (Q4) and quadratic quadrilateral (Q8) elements. The FEM results by using the three different elements are compared and their accuracies and efficiencies are discussed. One mesh plot and one stress contour plot are shown below.



An FE mesh (T6, 1518 elements)



FE stress plot and deformed shape (T6, 1518 elements)

The stress calculations with several meshes are listed in the following table, along with the number of elements and DOFs used.

Table. FEA stress results

Elem. Type	*No. of Elem.*	*Total DOFs*	*Max. σ (psi)*
Q4	506	1102	312.42
Q4	3352	7014	322.64
Q4	31349	64106	322.38
...	...	...	...
T6	1518	6254	320.18
T6	2562	10494	321.23
T6	24516	100702	322.24
...	...	...	...
Q8	501	3188	320.58
Q8	2167	13376	321.70
Q8	14333	88636	322.24

The converged results are obtained with all three types of elements with the differences in the maximum stress values less than 0.05%. However, Q8 and T6 elements are more efficient and converge much faster than the Q4 elements which is a linear representation and cannot model curved boundaries accurately. If the required accuracy is set at 1%, then the mesh with 501 Q8 elements should be sufficient. Note also that we need to check the deformed shape of the plate for each model to make sure the BCs are applied correctly. Less elements should be enough to achieve the same accuracy with a better or "smarter" mesh (mapped mesh). We will redo this example in the next chapter employing the symmetry features of the problem.

Further Discussions

(a) *Know the behaviors of each type of elements*:

T3 and Q4: linear displacement, constant strain and stress;

T6 and Q8: quadratic displacement, linear strain and stress.

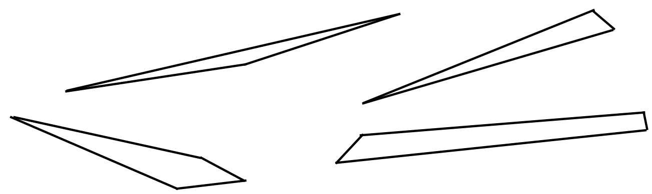
(b) *Choose the right type of elements for a given problem*:

When in doubt, use higher order elements (T6 or Q8) or a finer mesh.

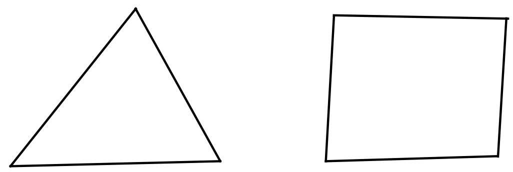
(c) *Avoid elements with large aspect ratios and corner angles* (Figure 3.14):

Aspect ratio = L_{max} / L_{min}

where L_{max} and L_{min} are the largest and smallest characteristic lengths of an element, respectively.



Elements with distorted shapes

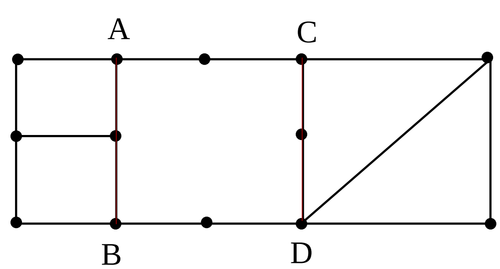


Elements with normal shapes

Figure 3.14. Elements with distorted (irregular) and normal (regular) shapes.

(d) Make sure the elements are connected properly:

Don not leave unintended gaps or free elements in FE models (Figure 3.15).



Improper connections (gaps along AB and CD)

Figure 3.15. Unintended gaps in the FE mesh.

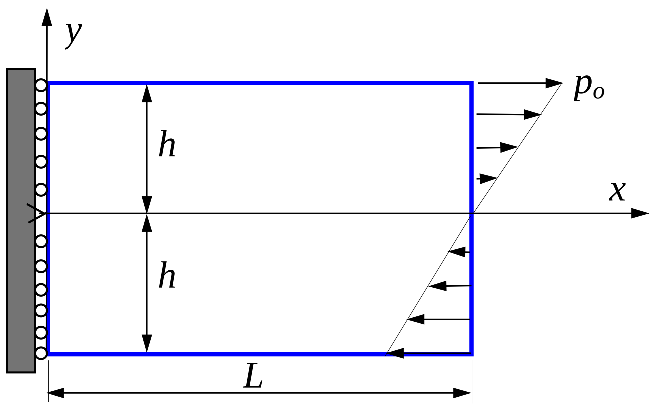
IV. Summary

2-D elements for analyzing plane stress and plane strain problems are discussed in this chapter. Linear triangular (T3) and linear quadrilateral (Q4) elements are good for deformation analysis and not accurate for stress analysis. Quadratic triangular (T6) and quadratic quadrilateral (Q8) elements are good for stress concentration problems and for models with curved boundaries. Whenever possible (as allowed by the computing resources), higher-order elements (T6 or Q8) elements should be applied in FE stress analysis of 2-D structures.

V. Problems

Problem 1. List the boundary conditions in Example 3.1.

Problem 2. The plate shown below is constrained at the left end and loaded with a linearly varying pressure load at the right end. Constants E, v and thickness t are given.



Suppose we have found the displacement field as follows:

$$u = \frac{p_0}{Eh} xy,$$

$$v = -\frac{p_0}{2Eh}(x^2 + \nu y^2)$$

Find:

(a) strains in the plate;

(b) stresses in the plate;

(c) check if or not the equilibrium equations are satisfied by the stresses;

(d) check if or not the boundary conditions are satisfied by the solution.

Optional:

Assume $E = 10\text{x}10^6$ psi, $v = 0.3$, $p_o = 100$ psi, $L = 12$ in., $h = 4$ in. and thickness $t = 0.1$ in. Use an FEM software to check your results.

Problem 3. Derive the shape functions in (3.17) for T3 elements and prove (3.18).

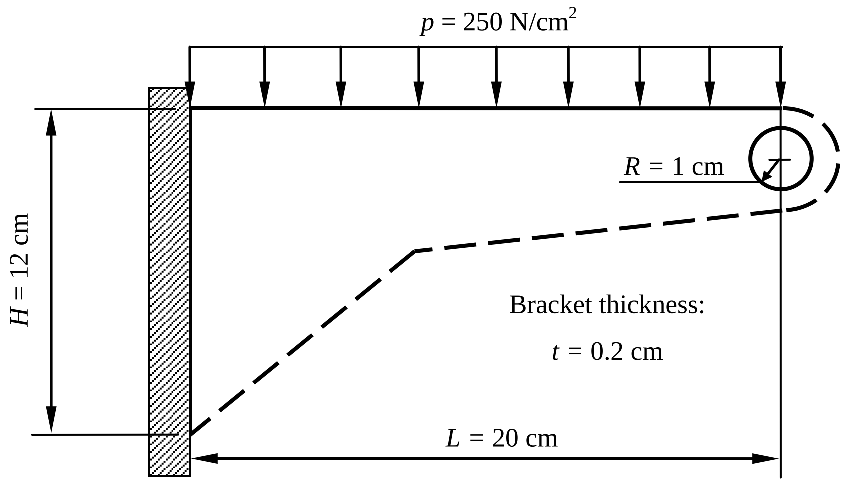
Problem 4. From (3.27), prove $\det \mathbf{J} = x_{13}y_{23} - x_{23}y_{13} = 2A$ and discuss why "bad shaped" elements can cause numerical errors in the FEM.

Problem 5. Using a FEM software, design a steel shelf bracket. Some dimensions of the bracket are fixed as shown in the figure, while others can be changed. The shape and topology near the lower part of the bracket can also be changed, including adding additional openings. The goal is to use as less material as possible for the bracket, while to support the given distributed load p.

(a) For steel, use E = 200 GPa, v = 0.32, and yield stress σ_Y = 250 MPa.

(b) Use a factor of safety = 2.0 for the design.

(c) Report the configuration, dimensions, and total volume of the bracket of your final design.

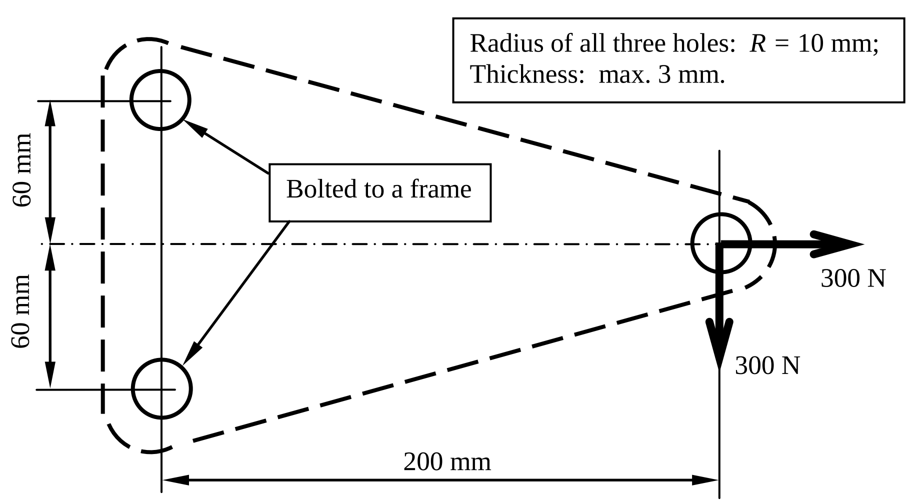


Problem 6. Similar to the previous problem, design a steel bracket. Some dimensions of the bracket are fixed as shown in the figure, while others can be changed. The shape and topology of the bracket can also be changed. The goal of this design is to use least material for the bracket, while to support the given loads.

(a) For steel, use E = 200 GPa, v = 0.32, and yield stress σ_Y = 250 MPa.

(b) Use a safety factor of 1.5 for the design.

(c) Report the configuration, all dimensions, and the *total volume* of the bracket of your final design.



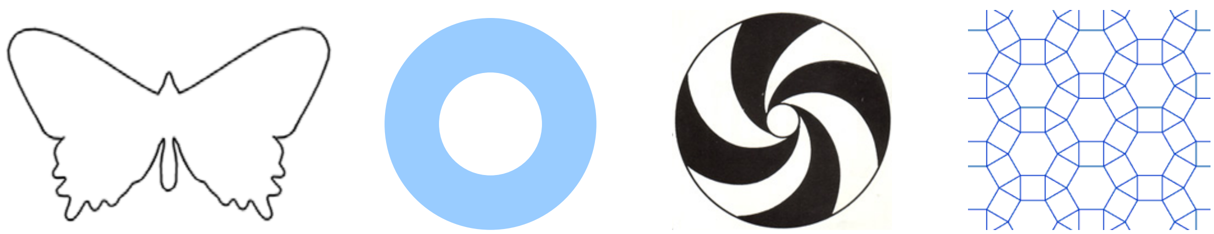
Chapter 4. Modeling and Solution Techniques

In this chapter, we discuss several techniques in the modeling and solution process of using the FEM. Applying these techniques can greatly improve the efficiencies and accuracies of the finite element analysis.

I. Symmetry

Symmetry features of a structure are the first thing one should look into and explore in the FE modeling and analysis. The model size can be cut almost in half and the solution efficiencies can be improved by several times. A structure possesses *symmetry* if its components are arranged in a periodic or reflective manner. Types of symmetries are (Figure 4.1):

- Reflective (mirror, bilateral) symmetry
- Axisymmetry
- Rotational (cyclic) symmetry
- Translational symmetry
- Others (or combinations of the above)



(a) (b) (c) (d)

Figure 4.1. Some examples of symmetry: (a) reflective symmetry; (b) axisymmetry; (c) rotational symmetry; and (d) translational symmetry.

In the FEM, symmetry properties can be applied to

- Reduce the size of the problems and thus save CPU time, disk space, post-processing efforts, and so on
- Simplify the modeling task
- Check the FEM results (make sure the results are symmetrical if the geometry and loading of the structure are symmetrical)

Symmetry properties of a structure should be fully exploited and retained in the FE model to ensure the efficiency and quality of FE solutions.

An Example

For the problem of a plate with a center hole as discussed in Example 3.2 of the previous chapter, we redo the FEA mesh using the symmetry features of the plate. To do this, we first model just one quarter of the plate using *mapped mesh*, and then reflect the model (with the mesh) twice to obtain the model and mesh for the entire plate, as shown in Figure 4.2. Only 896 Q8 elements are used in this symmetrical model and the results are comparable to those in Chapter 3 using more elements with the *free mesh*. The quarter model can also be applied in the analysis, if the boundary conditions are also symmetrical about the *xz* and *yz* planes.

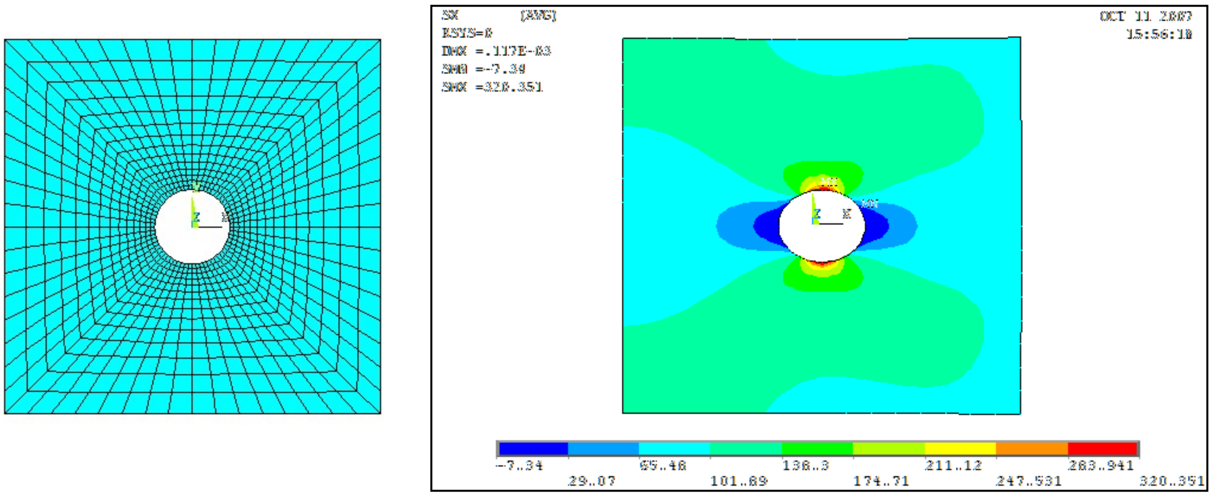


Figure 4.2. Results using symmetry features for Example 3.2 (mesh and stress contour plots).

In vibration or buckling analysis, however, the symmetry concept should not be used in the FEA solutions (it is still applicable in the modeling stage), since symmetric structures often have antisymmetric vibration or buckling modes.

II. Substructures (Superelements)

Another very useful technique for analyzing very large FEA models of mechanical systems is to apply the concept of substructures or superelements. Substructuring is a process of analyzing a large structure as a collection of (natural) components. The FEA models for these components are called *substructures* or *superelements* (SE). The physical meaning of a substructure is simply a finite element model of a portion of the structure. Mathematically, it presents a boundary matrix which is condensed by eliminating the *interior* points and keeping only the *exterior* or boundary points of the portion of the structure. In other words, instead of solving the FEA system of equations once, one can use partitions of the matrix so that larger models can be solved on relatively smaller computers. More details of the theory and implementations of the substructures or superelements can be found in the documentation of the FEA software packages (such as ANSYS or Nastran).

Figure 4.3 shows an FEA model of a truck used to conduct the full vehicle static or dynamic analysis. The entire model can have several millions of DOFs that can be beyond the capabilities of some computers. Using the substructuring technique, one can build the FEA model for each subsystem first (such as the cab, chassis, steering system, suspension system, payload, and so on) and then condense the FEA equations to smaller ones relating only DOFs on the interfaces between the subsystems and residing on a residual structure (e.g., the chassis). The condensed system is much smaller than the original system and can be solved readily.

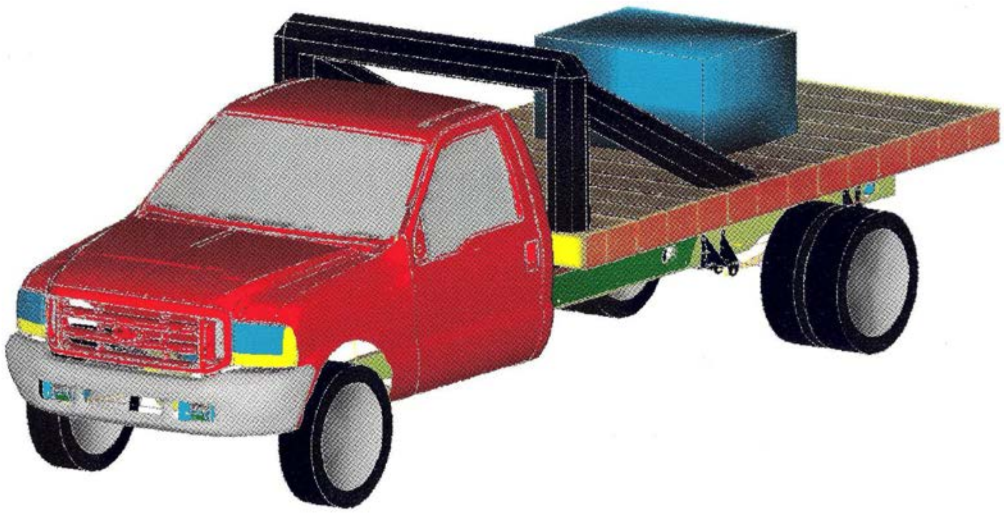


Figure 4.3. An FEA model of a truck analyzed using substructures.

The *advantages* of using the substructuring technique are:

- Good for large problems (which will otherwise exceed your computer capabilities)
- Less CPU time per run once the superelements have been processed (i.e., matrices have been condensed and saved)
- Components may be modeled by different groups
- Partial redesign requires only partial reanalysis (reduced cost)
- Efficient for problems with local nonlinearities (such as confined plastic deformations) which can be placed in one superelement (residual structure)
- Exact for static deformation and stress analysis

The *disadvantages* of using the substructuring technique are:

- Increased overhead for file management
- Increased initial time for setting up the system
- Matrix condensations for dynamic problems introduce new approximations

III. Equation Solving

There are two types of solvers used in the FEA for solving the linear systems of algebraic equations, mainly, the *direct* methods and *iterative* methods.

Direct Methods (Gauss Elimination):

- Solution time proportional to NB^2 (with N being the dimension of the matrix, B the bandwidth of the FEA systems)
- Suitable for small to medium problems (with DOFs in the 100,000 range), or slender structures (small bandwidth)
- Easy to handle multiple load cases

Iterative Methods:

- Solution time is unknown beforehand
- Reduced storage requirement
- Suitable for large problems, or bulky structures (large bandwidth, converge faster)
- Need to solve the system again for different load cases

An Example - Gauss Elimination:

Solve the following given system of equations:

$$\begin{bmatrix} 8 & -2 & 0 \\ -2 & 4 & -3 \\ 0 & -3 & 3 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 2 \\ -1 \\ 3 \end{Bmatrix} \qquad \text{or} \quad \mathbf{Ax} = \mathbf{b}\,. \tag{4.1}$$

Forward Elimination:

Form

$$\begin{array}{c} (1) \\ (2) \\ (3) \end{array} \left[\begin{array}{ccc:c} 8 & -2 & 0 & 2 \\ -2 & 4 & -3 & -1 \\ 0 & -3 & 3 & 3 \end{array}\right]; \tag{4.2}$$

(1) + 4 x (2) $\Rightarrow$ (2):

$$\begin{array}{c} (1) \\ (2) \\ (3) \end{array} \left[\begin{array}{ccc:c} 8 & -2 & 0 & 2 \\ 0 & 14 & -12 & -2 \\ 0 & -3 & 3 & 3 \end{array}\right]; \tag{4.3}$$

(2) + $\dfrac{14}{3}$(3) $\Rightarrow$ (3):

$$\begin{array}{c} (1) \\ (2) \\ (3) \end{array}\left[\begin{array}{ccc:c} 8 & -2 & 0 & 2 \\ 0 & 14 & -12 & -2 \\ 0 & 0 & 2 & 12 \end{array}\right]; \tag{4.4}$$

Back Substitutions (to obtain the solution):

$$\begin{aligned} x_3 &= 12/2 = 6 \\ x_2 &= (-2+12x_3)/14 = 5 \\ x_1 &= (2+2x_2)/8 = 1.5 \end{aligned} \qquad \text{or} \qquad \mathbf{x} = \begin{Bmatrix} 1.5 \\ 5 \\ 6 \end{Bmatrix}. \tag{4.5}$$

An Example - Iterative Method:

The Gauss-Seidel Method (as an example):

$$\mathbf{Ax} = \mathbf{b} \qquad (\mathbf{A} \text{ is symmetric}) \tag{4.6}$$

or

$$\sum_{j=1}^{N} a_{ij} x_j = b_i , \qquad i = 1, 2, \ldots, N.$$

Start with an estimate $\mathbf{x}^{(0)}$ of the solution vector and then iterate using the following:

$$x_i^{(k+1)} = \frac{1}{a_{ii}}\left[b_i - \sum_{j=1}^{i-1} a_{ij} x_j^{(k+1)} - \sum_{j=i+1}^{N} a_{ij} x_j^{(k)} \right], \qquad \text{for } i = 1, 2, \ldots, N. \tag{4.7}$$

In vector form,

$$\mathbf{x}^{(k+1)} = \mathbf{A}_D^{-1}\left[\mathbf{b} - \mathbf{A}_L \mathbf{x}^{(k+1)} - \mathbf{A}_L^T \mathbf{x}^{(k)}\right], \tag{4.8}$$

where

$\mathbf{A}_D = \langle a_{ii} \rangle$ is the diagonal matrix of $\mathbf{A}$,

$\mathbf{A}_L$ is the *lower triangular matrix* of $\mathbf{A}$,

such that

$$\mathbf{A} = \mathbf{A}_D + \mathbf{A}_L + \mathbf{A}_L^T. \tag{4.9}$$

Iterations continue until solution $\mathbf{x}$ converges, i.e.

$$\frac{\left\| \mathbf{x}^{(k+1)} - \mathbf{x}^{(k)} \right\|}{\left\| \mathbf{x}^{(k)} \right\|} \le \varepsilon , \tag{4.10}$$

where ε is the tolerance for convergence control.

Iterative solvers with moderate selections of the tolerance are usually much faster than direct solvers in solving large-scale models. However, for ill-conditioned systems, direct solvers should be applied to ensure the accuracy of the solutions.

IV. Nature of Finite Element Solutions

Some observation of the FEA models and solutions:

- FEA model – A mathematical model of the real structure, based on many approximations
- Real structure -- Infinite number of nodes (physical points or particles), thus infinite number of DOFs
- FEA model – finite number of nodes, thus finite number of DOFs

In particular, one can argue that the displacement field is controlled (or constrained) by the values at a limited number of nodes (Figure 4.4).

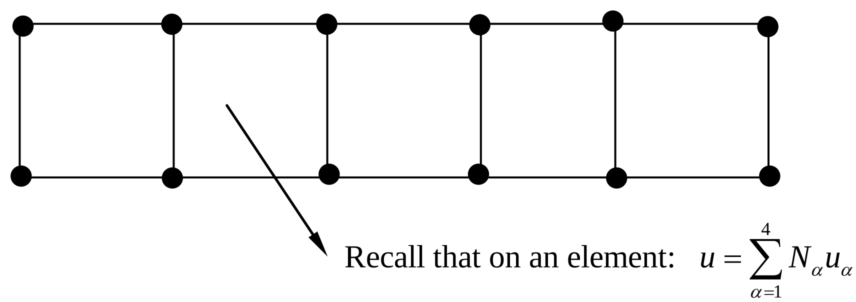


Figure 4.4. Elements in an FEA model

Therefore, we have the so called stiffening effect:

- FEA Model is stiffer than the real structure
- In general, displacement results are smaller in magnitudes than the exact values

Hence, the FEM solution of displacement is a *lower bound* of the exact solution.

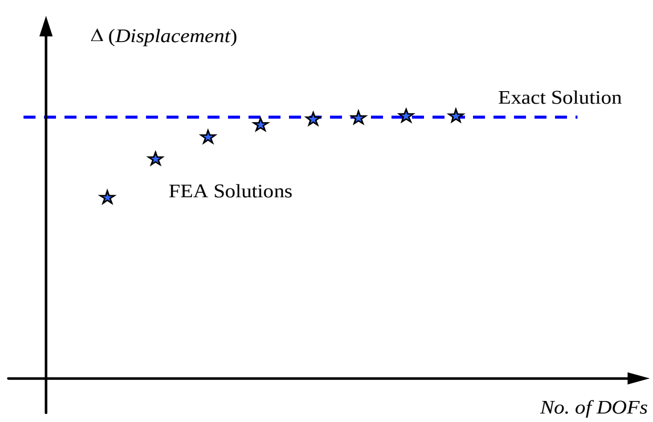


Figure 4.5. Convergence of FEM solutions with exact solution

That is, FEA displacement solutions approach the exact solution from below, which can be used to monitor the FEA solutions. However, this is true for the displacement based FEA.

V. Convergence of FEA Solutions

As the mesh in an FEA model is "refined" (with smaller and smaller elements) repeatedly, the FEA solution will converge to the exact solution of the mathematical model of the problem (the model based on bar, beam, plane stress/strain, plate, shell, or 3-D elasticity theories or assumptions). Several types of refinements have been devised in the FEA, which include:

h-refinement:	Reduce the size of the element ("*h*" refers to the typical size of the elements)
p-refinement:	Increase the order of the polynomials on an element (linear to quadratic, etc.; "*p*" refers to the highest order in a polynomial)
r-refinement:	Re-arrange the nodes in the mesh
hp-refinement:	Combination of the *h*- and *p*-refinements (to achieve better results)

With any of the above type of refinements, the FEA solutions will converge to the analytical solutions of the mathematical models. Some FEA software can automate the process of refinements in the FEA solutions to achieve the so called adaptive solutions.

VI. Adaptivity (*h*-, *p*-, and *hp*-Methods)

Adaptive FEA represents the future of the FEA applications. With proper error control, automatic refinements of an FEA mesh can be generated by the program until the converged FEA solutions are obtained. With the adaptive FEA capability, users' interactions are reduced, in the sense that a user only need to provide a good initial mesh for the model (even this step can be done by the software automatically).

Error estimates are crucial in the adaptive FEA. Interested readers can refer to Ref. [2] for more details. In the following, we introduce one type of the error estimates.

We first define two stress fields:

$\boldsymbol{\sigma}$ --- element by element stress field (discontinuous across elements)

$\boldsymbol{\sigma}^*$--- averaged or smoothed stress field (continuous across elements)

Then, the error stress field can be defined as:

$$\boldsymbol{\sigma}_E = \boldsymbol{\sigma} - \boldsymbol{\sigma}^* \tag{4.11}$$

Compute strain energies,

$$U = \sum_{i=1}^{M} U_i, \qquad U_i = \int_{V_i} \frac{1}{2} \boldsymbol{\sigma}^T \mathbf{E}^{-1} \boldsymbol{\sigma} \, dV \,; \tag{4.12}$$

$$U^* = \sum_{i=1}^{M} U_i^*, \qquad U_i^* = \int_{V_i} \frac{1}{2} \boldsymbol{\sigma}^{*T} \mathbf{E}^{-1} \boldsymbol{\sigma}^* \, dV \,; \tag{4.12}$$

$$U_E = \sum_{i=1}^{M} U_{Ei}, \qquad U_{Ei} = \int_{V_i} \frac{1}{2} \boldsymbol{\sigma}_E^T \mathbf{E}^{-1} \boldsymbol{\sigma}_E \, dV\,; \tag{4.14}$$

where M is the total number of elements, V_i is the volume of the element i.

One error indicator - the *relative energy error*, is defined as:

$$\eta = \left[\frac{U_E}{U + U_E} \right]^{1/2}. \qquad (0 \le \eta \le 1) \tag{4.15}$$

The indicator η is computed after each FEA solution. Refinement of the FEA model continues until, say

$$\eta \le 0.05.$$

When this condition is satisfied, we conclude that the converged FE solution is obtained.

Some examples of using different error estimates in the FEA solutions can be found in Ref. [2].

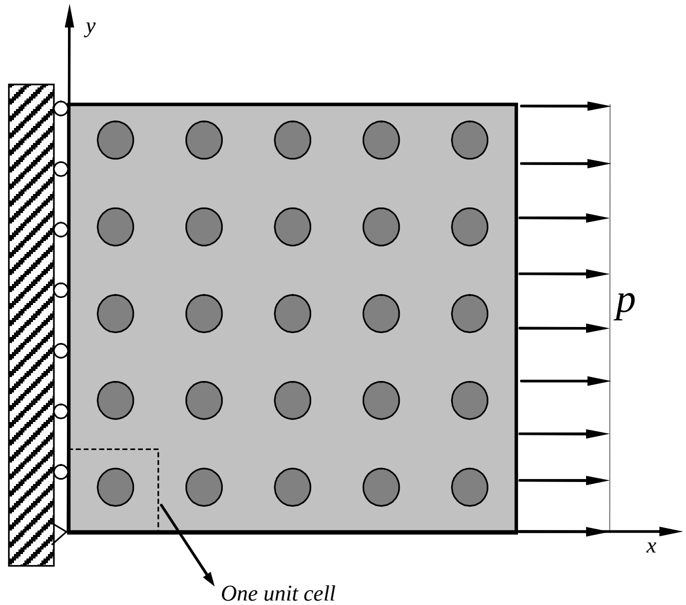
VII. Summary

In this chapter, we briefly discussed a few modeling techniques and concepts related to FEA solutions. For symmetrical structures, the symmetry features should be explored in both modeling/meshing stage and solution stage (if the BCs are symmetrical as well). Substructuring or using superelements is a useful technique for solving large-scale problems with constrained computing resources. Convergence of the FEA solutions is the important goal in FEA and should be monitored by using the error estimates and employing the adaptive FEA capabilities in the software.

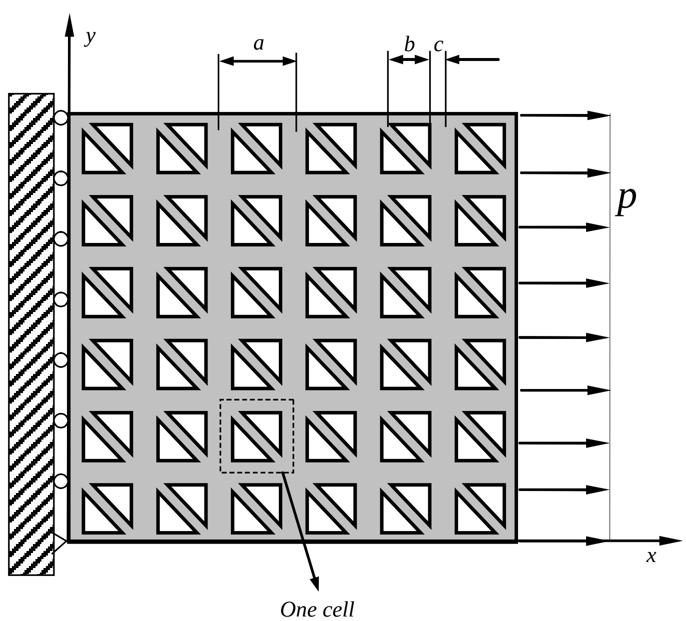
VIII. Problems

Problem 1. Suppose that we need to find out the in-plane effective modulus of a composite reinforced with long fibers aligned in the z-direction and distributed uniformly. A 2-D elasticity model shown below can be used for this study with the FEA. The effective modulus can be estimated by using the formula $E_{eff} = \sigma_{x(ave)} / \varepsilon_{x(ave)}$, where the averaged stress and strain are evaluated along the vertical edge on the right side of the model. Assume for the matrix E = 10 GPa, ν = 0.35, and for the fibers E = 100 GPa, ν = 0.3. The unit cell has a dimension of 1x1 μm^2, and the radius of the fibers is 0.2 μm.

Start with 1x1 cell, 2x2 cells, 3x3 cells, … and keep increasing the number of the cells as you can. Report the value of the effective Young's modulus of the composite in the x and y direction. Employ symmetry features of the model in generating the meshes for your analysis.



Problem 2. Suppose that a "meshed panel" will be used in a design in order to reduce the weight. For this purpose, we need to find out the in-plane effective modulus of this panel in the x- or y-direction. A sample piece of the panel similar to the one shown below can be used for this study. Employ symmetry and study the effects of the numbers of cells used in the model on the computed moduli.
Assume the panel is made of aluminum with E = 70 GPa, ν = 0.35, a = 10 mm, b = 6 mm, c = 1.5 mm, and thickness t = 1 mm.



Chapter 5. Plate and Shell Elements

Many structure members can be categorized as plates and shells [11], which are extensions of the 1-D straight beams and curved beams to 2-D cases, respectively. Some of the structures that can be modeled as plates are:

- Shear walls
- Floor panels
- Shelves

While those that can be modeled as shells include:

- Sea shells and egg shells (the wonder of the nature)
- Various containers, pipes, and tanks
- Bodies of cars, boats, aircraft, etc.
- Roofs of buildings (the Superdome), etc.

Figure 5.1 shows two recent engineering wonders that are constructed mainly using plate and shell structure members.

(a) The new Boeing 787 aircraft   (b) The National Grand Theatre in Beijing

Figure 5.1. Examples of plate and shell structures.

The advantages of using plate and shell structures are their light weight, superior load-carrying capabilities, and sometimes, simply their artistic appeals.

I. Plate Theory

A plate has the following characteristics:

- A flat surface
- Applied with lateral loading
- Bending behavior dominates

Forces and Moments Acting on the Plate

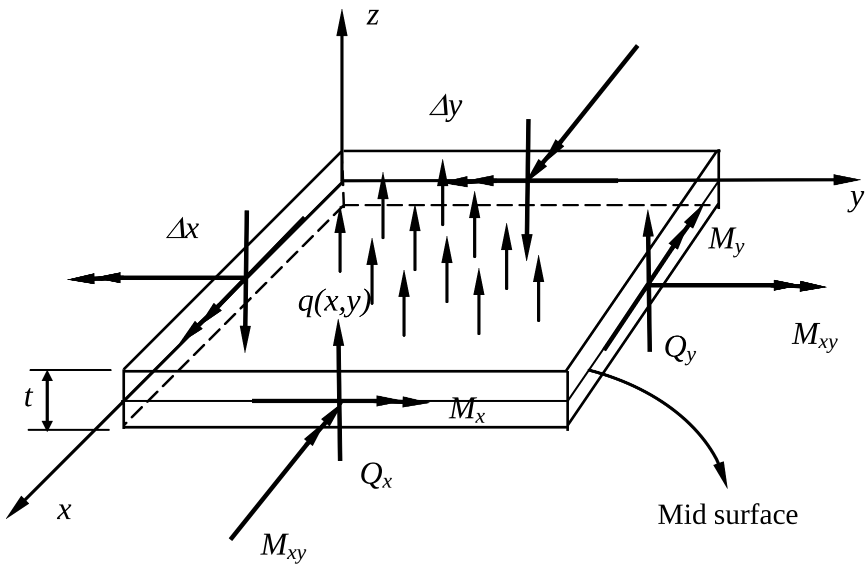


Figure 5.2. Forces and moments acting on an infinitesimally small element in a plate.

Stresses in the Plate

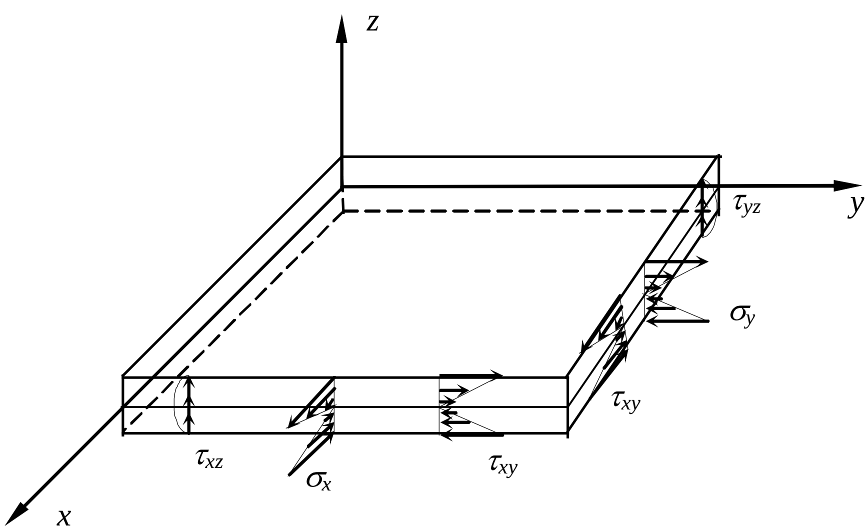


Figure 5.3. Stresses acting on the infinitesimally small element in the plate.

Relations Between the Forces and Stresses:

Bending moments (per unit length):

$$M_x = \int_{-t/2}^{t/2} \sigma_x z dz, \qquad (N \cdot m / m) \qquad (5.1)$$

$$M_y = \int_{-t/2}^{t/2} \sigma_y z dz, \qquad (N \cdot m / m) \qquad (5.2)$$

Twisting moment (per unit length):

$$M_{xy} = \int_{-t/2}^{t/2} \tau_{xy} z dz, \qquad (N \cdot m / m) \qquad (5.3)$$

Shear Forces (per unit length):

$$Q_x = \int_{-t/2}^{t/2} \tau_{xz} dz, \qquad (N / m) \qquad (5.4)$$

$$Q_y = \int_{-t/2}^{t/2} \tau_{yz} dz, \qquad (N / m) \qquad (5.5)$$

Maximum bending stresses:

$$(\sigma_x)_{\max} = \pm \frac{6M_x}{t^2}, \qquad (\sigma_y)_{\max} = \pm \frac{6M_y}{t^2}. \qquad (5.6)$$

Note that:

- Maximum stress is always at $z = \pm t/2$
- No bending stresses at midsurface (similar to the beam model)

Thin Plate Theory (Kirchhoff Plate Theory)

Assumptions (similar to those for the simple beam theory):

A straight line along the normal to the mid surface remains straight and normal to the deflected mid surface after loading, that is, there is no transverse shear deformation (Figure 5.4):

$$\gamma_{xz} = \gamma_{yz} = 0 .$$

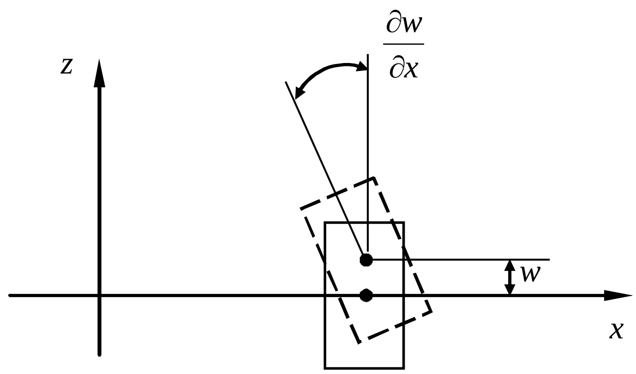


Figure 5.4. Deflection and rotation after loading of a plate according to Kirchhoff plate theory.

Displacement:

$$\begin{aligned} w &= w(x, y), \qquad (\textit{deflection}) \\ u &= -z \frac{\partial w}{\partial x}, \\ v &= -z \frac{\partial w}{\partial y}. \end{aligned} \tag{5.7}$$

Strains:

$$\begin{aligned} \varepsilon_x &= -z \frac{\partial^2 w}{\partial x^2}, \\ \varepsilon_y &= -z \frac{\partial^2 w}{\partial y^2}, \\ \gamma_{xy} &= -2z \frac{\partial^2 w}{\partial x \partial y}. \end{aligned} \tag{5.8}$$

Note that there is no stretch of the mid surface due to the deflection of the plate.

Stresses (plane stress state):

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}, \tag{5.9}$$

or,

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = -z\frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu) \end{bmatrix} \begin{Bmatrix} \dfrac{\partial^2 w}{\partial x^2} \\ \dfrac{\partial^2 w}{\partial y^2} \\ \dfrac{\partial^2 w}{\partial x \partial y} \end{Bmatrix}. \tag{5.10}$$

Note the main variable: deflection $w = w(x, y)$.

Governing Equation:

$$D\nabla^4 w = q(x, y), \tag{5.11}$$

where

$$\nabla^4 \equiv \left(\frac{\partial^4}{\partial x^4} + 2\frac{\partial^4}{\partial x^2 \partial y^2} + \frac{\partial^4}{\partial y^4}\right),$$

$$D = \frac{Et^3}{12(1-\nu^2)} \quad \text{(the bending rigidity of the plate)}, \tag{5.12}$$

q = lateral distributed load (force/area).

Compare the 1-D equation for straight beam:

$$EI\frac{d^4 w}{dx^4} = q(x). \tag{5.13}$$

Note: Equation (5.11) represents the equilibrium condition in the z-direction. To see this, refer to the Figure 5.2. showing all the forces on a plate element. Summing the forces in the z-direction, we have,

$$Q_x \Delta y + Q_y \Delta x + q\Delta x \Delta y = 0, \tag{5.14}$$

which yields,

$$\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} + q(x,y) = 0\,. \tag{5.15}$$

Substituting the following relations into the above equation, we obtain Eq. (5.11):

Shear forces and bending moments:

$$Q_x = \frac{\partial M_x}{\partial x} + \frac{\partial M_{xy}}{\partial y}, \qquad Q_y = \frac{\partial M_{xy}}{\partial x} + \frac{\partial M_y}{\partial y}, \tag{5.16}$$

$$M_x = D\left(\frac{\partial^2 w}{\partial x^2} + \nu\frac{\partial^2 w}{\partial y^2}\right), \qquad M_y = D\left(\frac{\partial^2 w}{\partial y^2} + \nu\frac{\partial^2 w}{\partial x^2}\right). \tag{5.17}$$

The fourth-order partial differential equation, given in (5.11) and in terms of the deflection $w(x,y)$, needs to be solved under certain given boundary conditions.

Boundary Conditions:

Clamped: $\quad w = 0, \quad \dfrac{\partial w}{\partial n} = 0\,;$ (5.18)

Simply supported: $\quad w = 0, \quad M_n = 0\,;$ (5.19)

Free: $\quad Q_n = 0, \quad M_n = 0\,;$ (5.20)

where n is the normal direction of the boundary (Figure 5.5). Note that the given values in the boundary conditions shown above can be non-zero values as well.

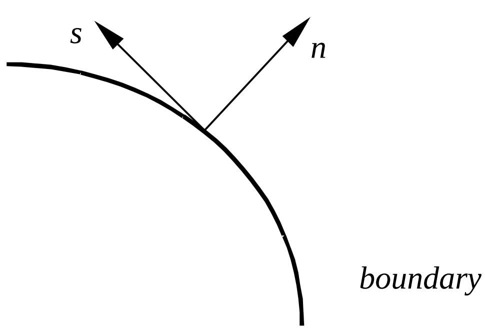


Figure 5.5. The boundary of a plate.

Examples:

A square plate (Figure 5.6) with four edges clamped or hinged, and under a uniform load q or a concentrated force P at the center C.

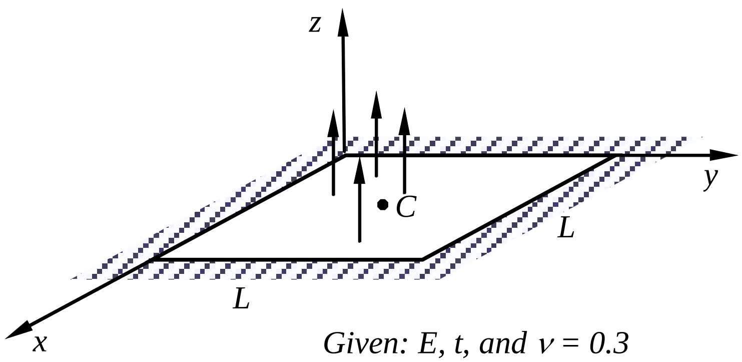


Figure 5.6. A square plate.

For this simple geometry, Eq. (5.11) with boundary condition (5.18) or (5.19) can be solved analytically. The maximum deflections are given in the Table 5.1 for the four different cases.

Table 5.1. Deflection at the Center (w_c)

	Clamped	*Simply supported*
Under uniform load q	$0.00126\ qL^4/D$	$0.00406\ qL^4/D$
Under concentrated force P	$0.00560\ PL^2/D$	$0.0116\ PL^2/D$

in which: $D = Et^3/(12(1-\nu^2))$.

These values can be used to verify the FEA solutions.

Thick Plate Theory (Mindlin Plate Theory)

If the thickness t of a plate is not "thin", for example, when $t/L \geq 1/10$ (L = a characteristic dimension of the plate main surface), then the thick plate theory by Mindlin should be applied. This theory accounts for the angle changes within a cross section, that is,

$$\gamma_{xz} \neq 0, \qquad \gamma_{yz} \neq 0\,.$$

This means that a line which is normal to the mid surface before the deformation will not be so after the deformation (Figure 5.7).

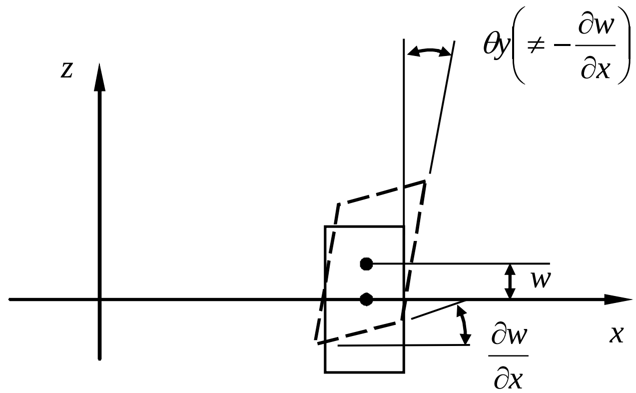


Figure 5.7. Displacement and rotation based on the Mindlin thick plate theory.

New independent variables:

θ_x and θ_y: rotation angles of a line, which is normal to the mid surface before the deformation, about *x*- and *y*-axis, respectively.

New relations:

$$u = z\theta_y, \qquad v = -z\theta_x; \tag{5.21}$$

and

$$\begin{aligned} &\varepsilon_x = z\frac{\partial \theta_y}{\partial x}, \quad \varepsilon_y = -z\frac{\partial \theta_x}{\partial y}, \quad \gamma_{xy} = z(\frac{\partial \theta_y}{\partial y} - \frac{\partial \theta_x}{\partial x}), \\ &\gamma_{xz} = \frac{\partial w}{\partial x} + \theta_y, \quad \gamma_{yz} = \frac{\partial w}{\partial y} - \theta_x. \end{aligned} \tag{5.22}$$

Note that if we imposed the conditions (or assumptions) that

$$\gamma_{xz} = \frac{\partial w}{\partial x} + \theta_y = 0, \qquad \gamma_{yz} = \frac{\partial w}{\partial y} - \theta_x = 0, \tag{5.23}$$

then we can recover the relations applied in the thin plate theory.

Main variables are: $w(x,y)$, $\theta_x(x,y)$ and $\theta_y(x,y)$.

The governing equations and boundary conditions can be established for thick plates based on the above assumptions.

II. Plate Elements

Kirchhoff Plate Elements:

A 4-Node Quadrilateral Element:

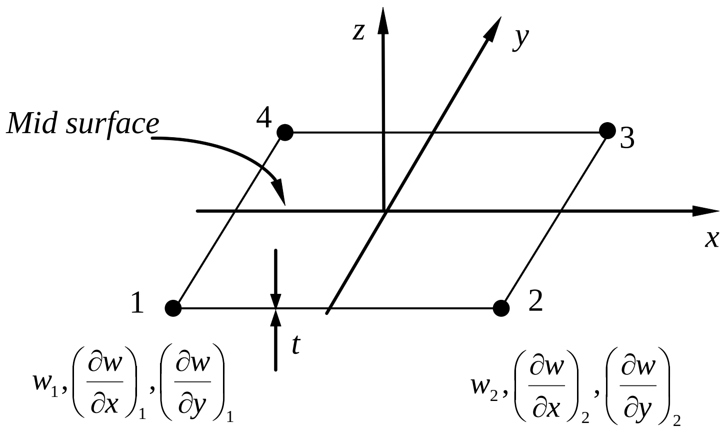


Figure 5.8. A 4-node quadrilateral element with 3 DOFs at each node.

DOFs at each node: $w, \dfrac{\partial w}{\partial x}, \dfrac{\partial w}{\partial y}.$

On each element, the deflection *w(x,y)* is represented by

$$w(x,y) = \sum_{i=1}^{4} \left[N_i w_i + N_{xi} \left(\frac{\partial w}{\partial x}\right)_i + N_{yi} \left(\frac{\partial w}{\partial y}\right)_i \right], \tag{5.24}$$

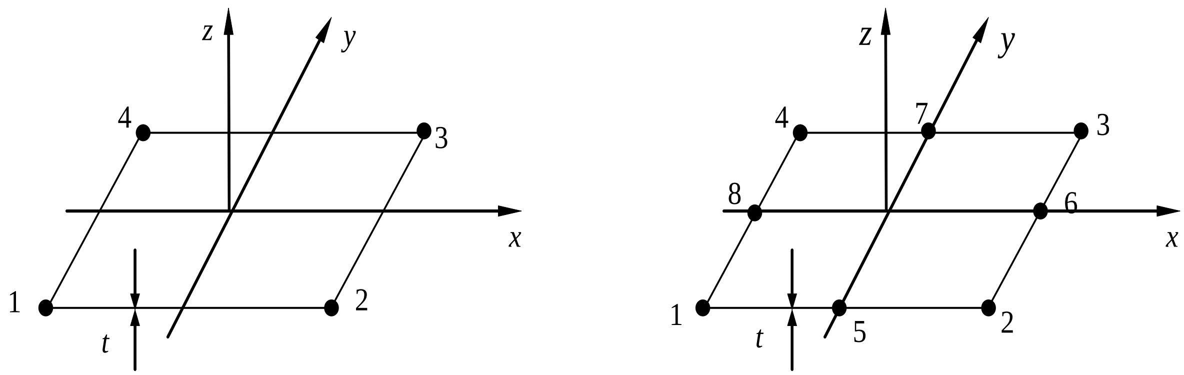
where N_i, N_{xi} and N_{yi} are shape functions. This is an incompatible element [4]. The stiffness matrix is still of the form

$$\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E} \mathbf{B} dV, \tag{5.25}$$

where **B** is the strain-displacement matrix, and **E** the Young's modulus (stress-strain) matrix.

Mindlin Plate Elements:

Two Quadrilateral Elements:



(a) 4-node quadrilateral (b) 8-node quadrilateral

Figure 5.9. 4-node and 8-node quadrilateral plate elements.

DOFs at each node: w, θ_x and θ_y.

On each element, the displacement and rotations are represented by:

$$
\begin{aligned}
w(x, y) &= \sum_{i=1}^{n} N_i w_i, \\
\theta_x(x, y) &= \sum_{i=1}^{n} N_i \theta_{xi}, \\
\theta_y(x, y) &= \sum_{i=1}^{n} N_i \theta_{yi}.
\end{aligned}
\qquad (5.26)
$$

For these elements:

- There are three independent fields within each element.
- Deflection $w(x, y)$ is linear for Q4, and quadratic for Q8.

Discrete Kirchhoff Element:

This is a triangular element. First, start with a 6-node triangular element (Figure 5.10),

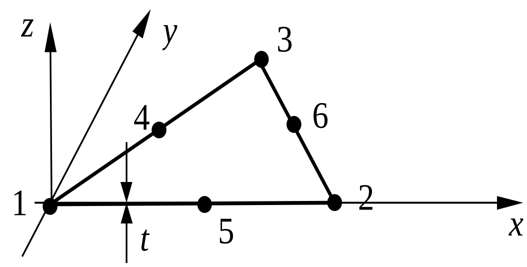


Figure 5.10. A 6-node triangular element with 5 DOFs at each corner node and 2 DOFs at each mid node.

DOFs at corner nodes: $w, \ \dfrac{\partial w}{\partial x}, \ \dfrac{\partial w}{\partial y}, \ \theta_x, \ \theta_y;$

DOFs at mid side nodes: $\theta_x, \ \theta_y.$

Total DOFs = 21.

Then, impose conditions $\gamma_{xz} = \gamma_{yz} = 0$, etc., at selected nodes to reduce the DOFs (using relations in Eq. (5.22)), to obtain the discrete Kirchhoff triangular (DKT) element (Figure 5.11):

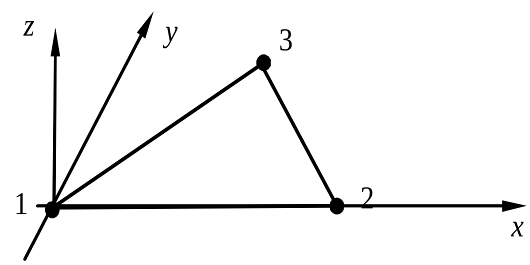


Figure 5.11. Discrete Kirchhoff triangular element with 3 DOFs at each node.

DOFs at each node: $w, \theta_x \left(= \dfrac{\partial w}{\partial x} \right), \theta_y \left(= \dfrac{\partial w}{\partial y} \right).$

Total DOFs = 9 (DKT element).

Note that $w(x, y)$ is incompatible for DKT elements [4]; however, its convergence is faster (w is cubic along each edge) and it is efficient.

Test Problem:

We consider a square plate with its four edges clamped and a concentrated force P applied at the center (Figure 5.12). Using 4-node plate elements, we obtain results in Table 5.2.

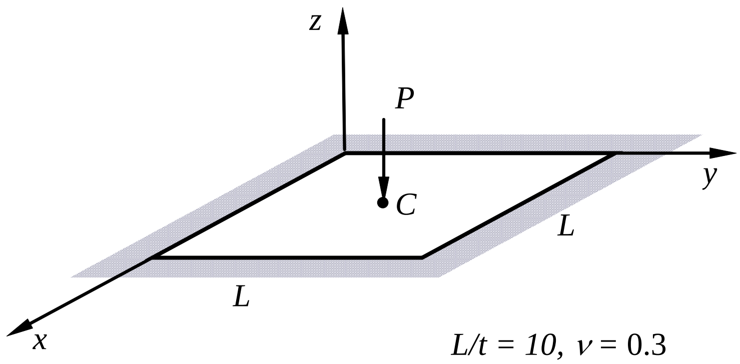


Figure 5.12. A clamped square plate with a concentrated load P.

Table 5.2. ANSYS result for deflection w_c.

Number of Elements	w_c ($\times PL^2/D$)
2×2	0.00593
4×4	0.00598
8×8	0.00574
16×16	0.00565
:	:
Exact Solution	0.00560

Questions: Why results converge from "above"? Contradiction to what we learnt about the nature of the FEA solution?

Reason: This is an incompatible element (See comments on page 177 of Cook's textbook [4]).

III. Shells and Shell Elements

Shells are thin structure members which span over curved surfaces. The thickness *t* of a shell is usually much smaller than the other dimensions of the shell and thus it can be represented mathematically by a 2-D surface in space, with the thickness as a parameter (Figure 5.13).

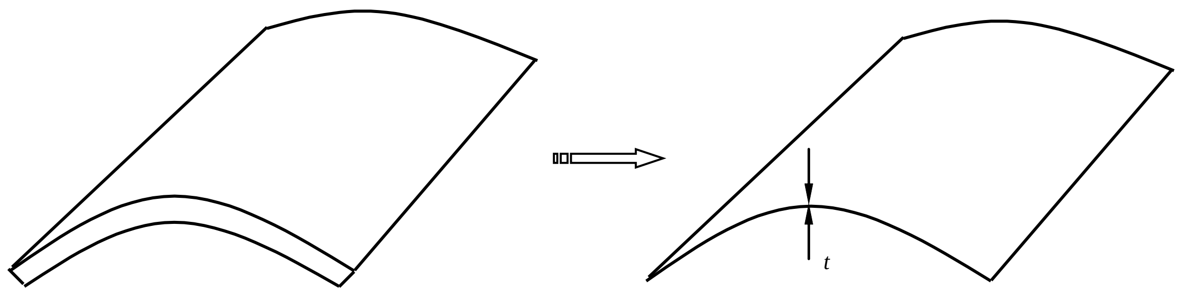


Figure 5.13. A shell structure member and its mathematical representation.

Forces in Shells:

Unlike the plate models, there are two types of forces in shells, that is:

Membrane forces (in plane forces) + Bending forces (out of plane forces)

(cf. plates: bending forces only)

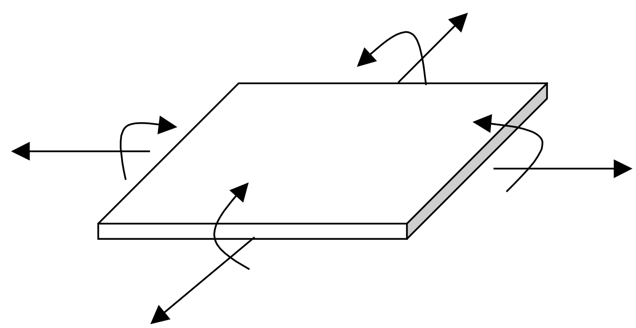
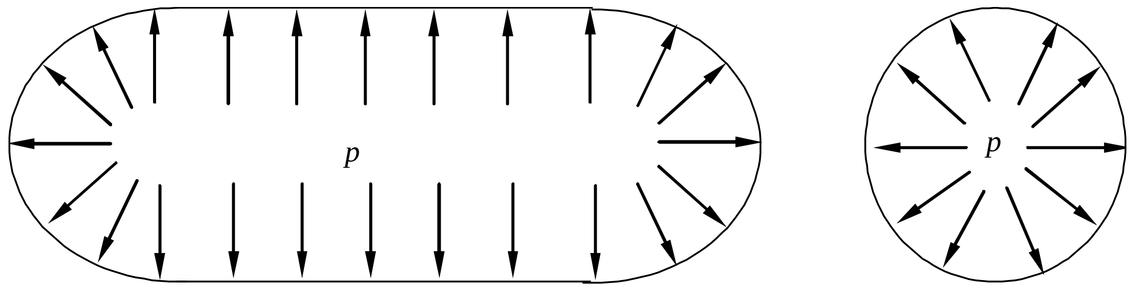


Figure 5.14. Forces and moments in a shell structure member.

Example: A Cylindrical Container



internal forces:

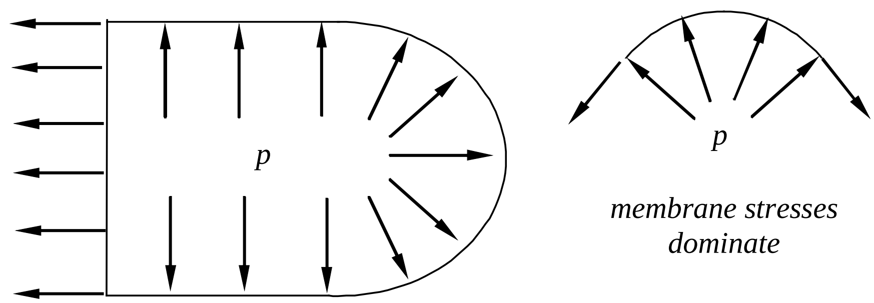


Figure 5.15. Internal forces acting in a cylinder under internal pressure load.

Shell Theories:

Similar to the plate theories, there are two types of theories for modeling shells, according to the thickness of the shells to be studied:

- Thin shell theory
- Thick shell theory

Shell theories are the most complicated ones to formulate and analyze in mechanics. Many of the contributions were made by Russian scientists in the 1940s and 1950s, due to the need to develop new aircraft and other light-weight structures. Interested readers can refer to Ref. [11] for in-depth studies on this subject. These theoretical work have laid the foundations for the development of various finite elements for analyzing shell structures.

Flat Shell Elements:

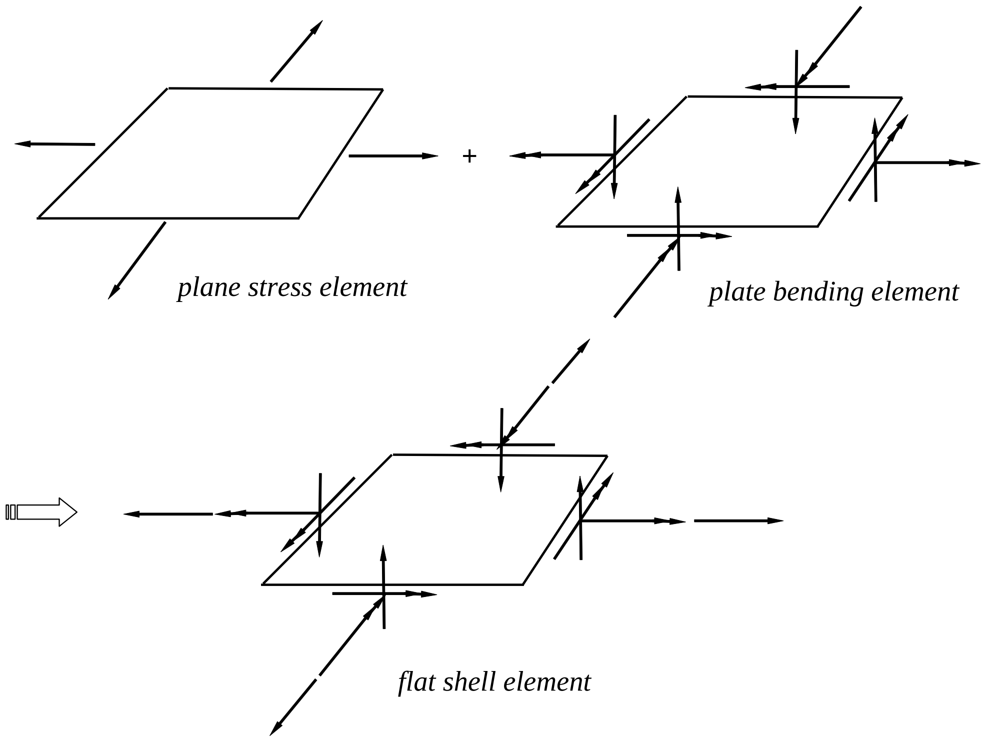


Figure 5.16. Combination of plane stress element and plate bending element yields a flat shell element.

cf.: bar + simple beam element => general beam element (for modeling curved beams).

DOFs at each node:

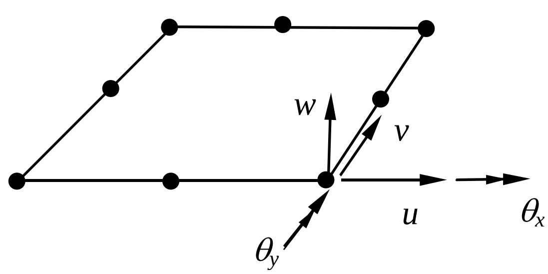


Figure 5.17. Q4 or Q8 shell elements.

Curved Shell Elements:

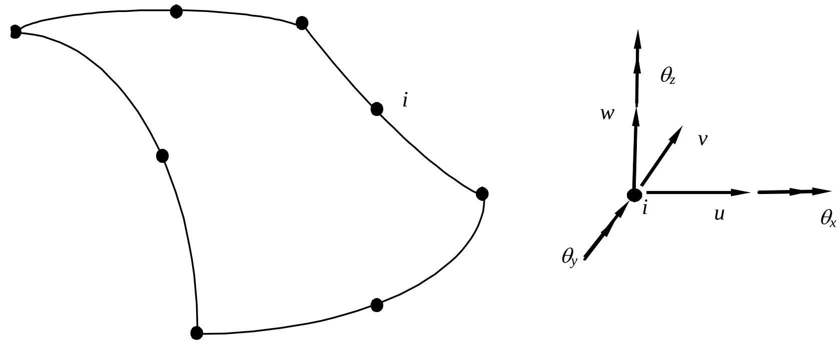


Figure 5.18. A 8-node curved shell element and the DOFs at a typical node *i*.

- Curved shell elements are based on the various shell theories;
- They are the most general shell elements (flat shell and plate elements are subsets);
- Complicated in formulation.

Test Cases:

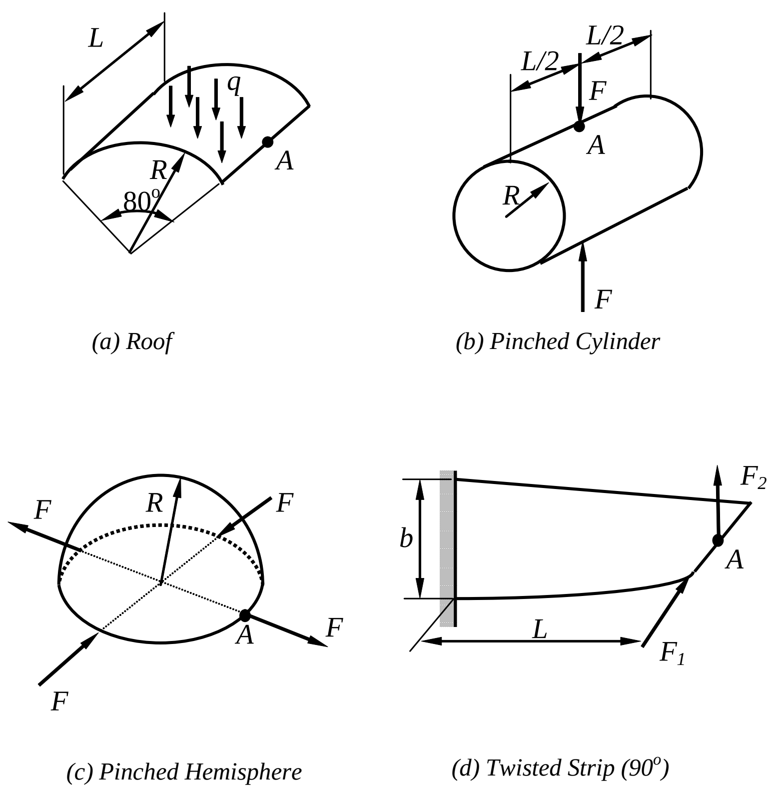


Figure 5.19. Four test cases where analytical solutions are available.

For the four cases shown in Figure 5.19, check the table on page 188 of Cook's textbook [4] for the exact values of displacement Δ_A under various loading conditions, which can be used to verify the FEA results in shell analysis.

Figure 5.20 is a stamping part analyzed using shell elements. The bracket has a uniform thickness and is fixed at the four bolt hole positions. A load is applied through a pin passing through the two holes in the lower part of the bracket. Note the one layer of elements on the edge of each hole (Figure 5.20 (a)), which is a common practice to models holes. Note also that this layer of elements on the edge of each hole has been masked in the stress contour plot (Figure 5.20 (b)), due to inaccurate stress results near the constraint locations. To reduce the true stress levels in the bracket, the thickness can be changed, the shape of the bracket can be modified, and the model is re-meshed and re-analyzed, all of which are very easy to carry out with the shell elements.

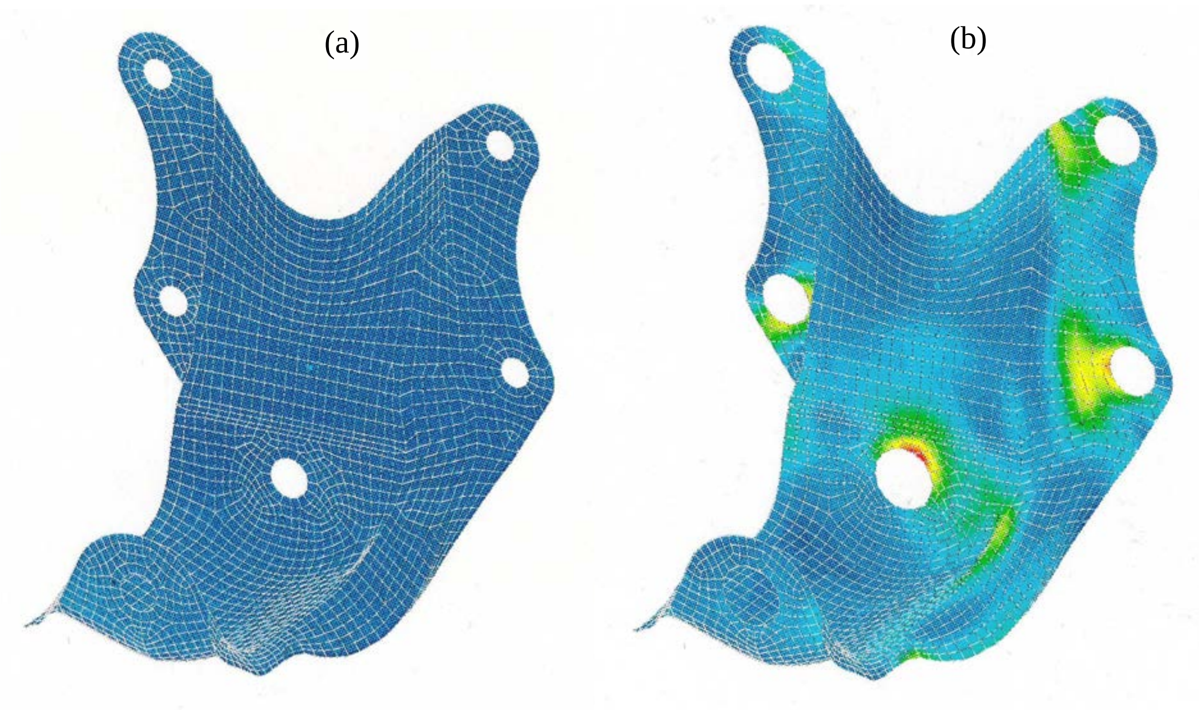


Figure 5.20. Stress analysis of a bracket using shell elements: (a) The FEA model; (b) Stress contour plot.

Cautions in Applying Shell Elements:

In many cases, however, the plate and shell models may not be adequate for analyzing a structure member, even if it is considered thin. For example, the structure component has a nonuniform thickness (turbine blades, vessels with stiffeners, thin layered structures, etc.), see Figure 5.21, or has a crack for which detailed stress analysis is needed. In such cases, one should turn to 3-D elasticity theory and apply solid elements which will be discussed in the next chapter.

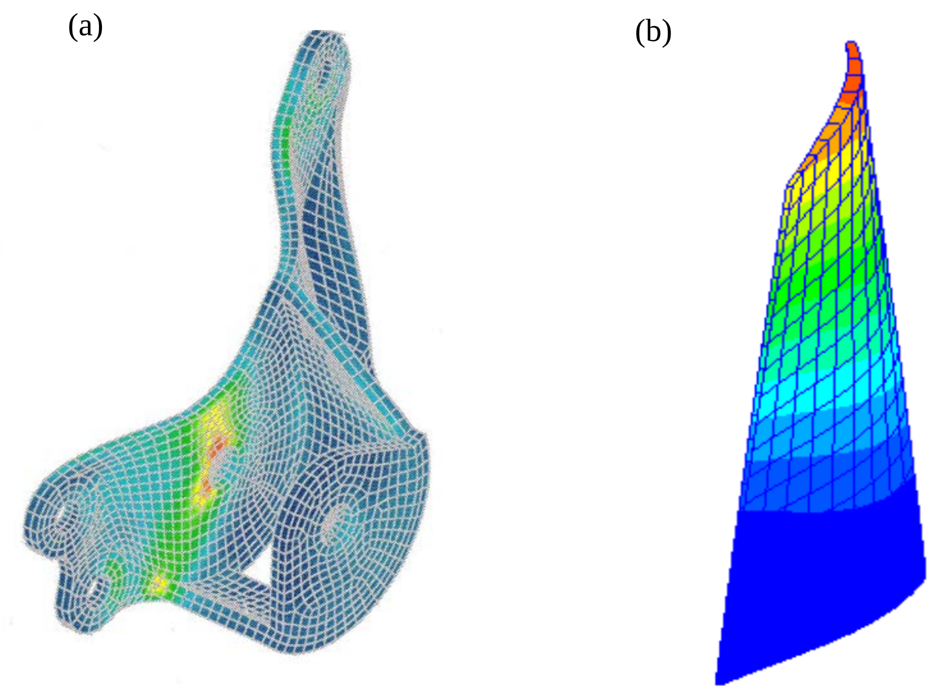


Figure 5.21. Cases in which shell elements are *not* adequate: (a) Casting parts; (b) Parts with nonuniform thickness. 3-D solid elements should be applied in such cases.

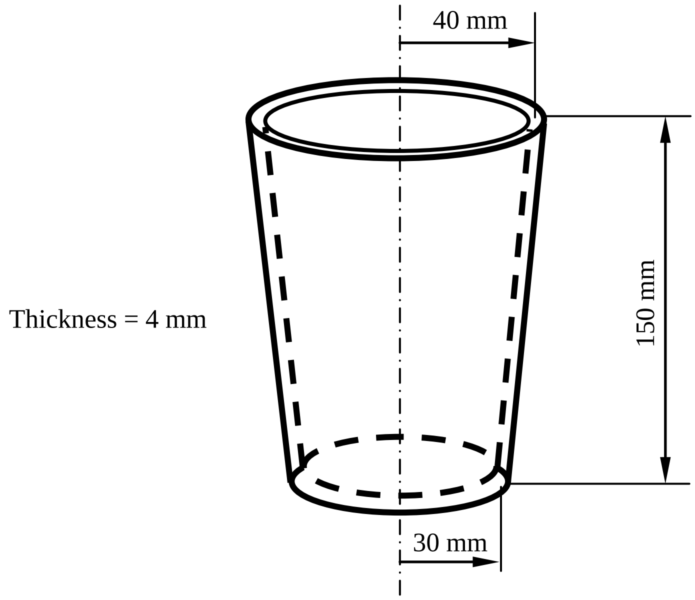
IV. Summary

In this chapter, we discussed the main aspects of the plate and shell theories and the plate and shell elements used for analyzing plate and shell structures. Plates and shells can be regarded as the extensions of the beam elements from 1-D line elements to 2-D surface elements. Plates are usually applied in modeling flat thin structure members, while the shells in modeling curved thin structure members. In applying the plate and shell elements, one should keep in mind the assumptions used in the development of these types of elements. In cases where these assumptions are no longer valid, one should turn to general 3-D theories and solid elements.

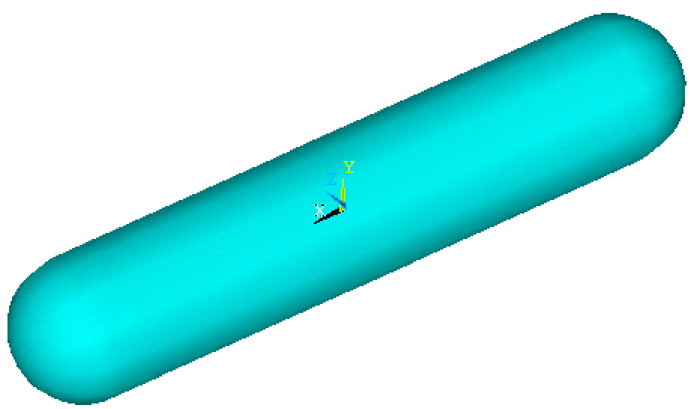
V. Problems

Problem 1. The roof structure shown in Figure 5.19 (a) is loaded by its own weight with q = 90 $\text{lb}_\text{f}/\text{in}^2$. The dimensions and material constants are: R = 25 in., L = 50 in., t = 0.25 in., $E = 432\times10^6$ psi and ν = 0.0. The two straight edges are free, while the two curved edges have a “diaphragm” support (meaning that y and z DOFs are constrained, but x (along the axis) and all rotational DOFs are unconstrained). Use shell elements to find the maximum displacement and von Mises stress in the structure. Verify your results (Note that the value of the analytical solution for the displacement at the mid point A of the straight edge is 0.3024 in).

Problem 2. Consider a glass cup placed on a table, as shown in the figure. Using an FEA software package and 8-node shell elements, find the maximum displacement and von Mises stress in the cup when the cup is applied with a pressure load of 10 N/mm^2 on the inner wall. Assume that the cup has a uniform thickness, E = 70 GPa and v = 0.17.



Problem 3. A fuel tank, with a total length = 5 m, diameter = 1 m, and thickness = 0.01 m, is shown below. Using the FEA, find the deformation and stresses when the tank is applied with an internal pressure p = 100 MPa and placed on the ground.



Assume the Young's modulus E = 200 GPa and Poisson's ratio υ = 0.3.

Chapter 6. Three-Dimensional Elasticity Problems

Solid elements based on 3-D elasticity theory [9, 10] are the most general elements for stress analysis when the simplified bar, beam, plane stress/strain, plate/shell elements are no longer valid or accurate. In this chapter, we first review the elasticity equations in 3-D and then discuss a few types of 3-D finite elements commonly used for 3-D stress analysis.

I. 3-D Elasticity Theory

Stress State:

There are six stress components at each point in a 3-D elastic body (Figure 6.1).

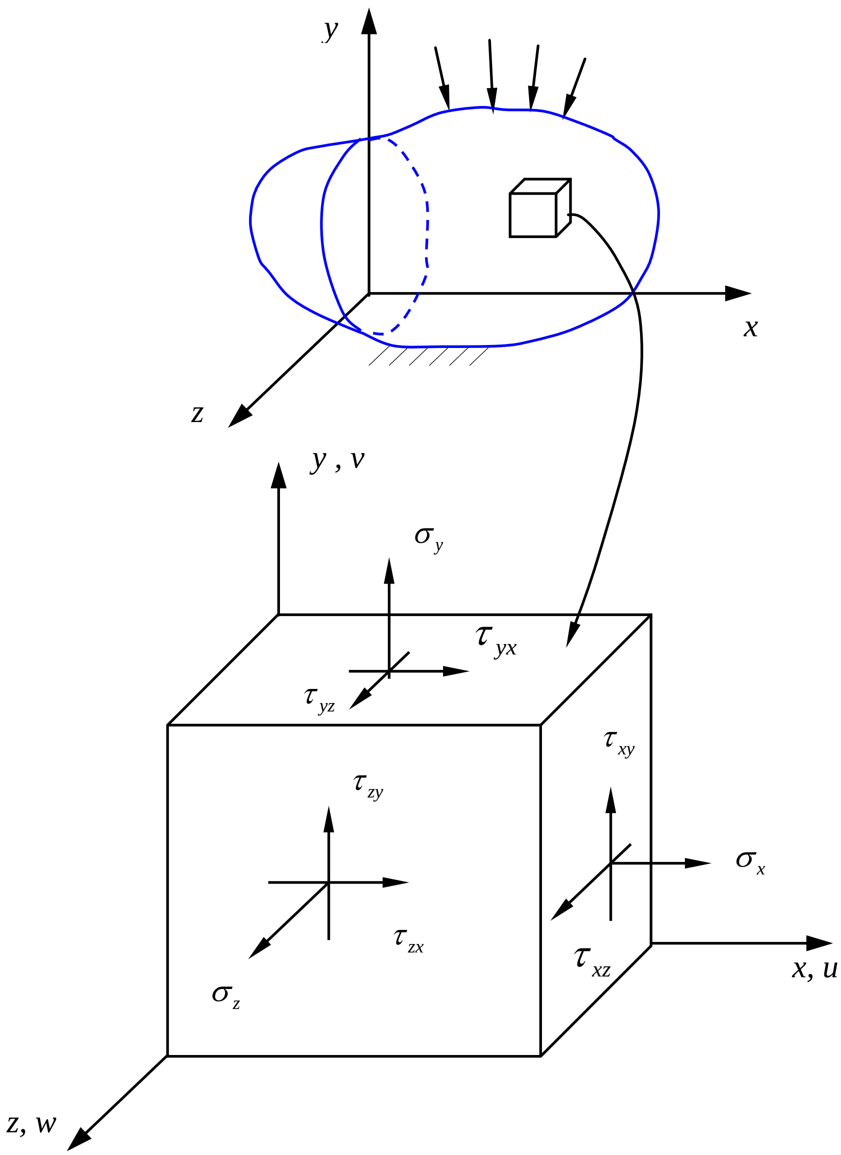


Figure 6.1. State of stress in a 3-D element.

The six stress components shown in Figure 6.1 can be wriiten as a vector:

$$\boldsymbol{\sigma} = \{\sigma\} = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}, \qquad \text{or} \quad [\sigma_{ij}]. \tag{6.1}$$

Strains:

Similarly, the six strain components in 3-D can be expressed as:

$$\boldsymbol{\varepsilon} = \{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}, \qquad \text{or} \quad [\varepsilon_{ij}]. \tag{6.2}$$

Stress-Strain Relations:

The stress-strain relation in 3-D is given by:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \frac{E}{(1+v)(1-2v)} \begin{bmatrix} 1-v & v & v & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2v}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2v}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2v}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}. \tag{6.3}$$

Or in a matrix form:

$$\boldsymbol{\sigma} = \mathbf{E}\boldsymbol{\varepsilon}.$$

Displacements:

The displacement field can be described as:

$$\mathbf{u} = \begin{Bmatrix} u(x,y,z) \\ v(x,y,z) \\ w(x,y,z) \end{Bmatrix} = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}. \tag{6.4}$$

Strain-Displacement Relations:

Strain field is related to the displacement field as given below:

$$\varepsilon_x = \frac{\partial u}{\partial x}, \quad \varepsilon_y = \frac{\partial v}{\partial y}, \quad \varepsilon_z = \frac{\partial w}{\partial z},$$
$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}, \quad \gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}, \quad \gamma_{xz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}. \tag{6.5}$$

These six equations can be written in the following index or tensor form:

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right), \qquad i,\ j = 1, 2, 3.$$

Or simply,

$$\varepsilon_{ij} = \frac{1}{2}\left(u_{i,j} + u_{j,i}\right). \qquad \text{(tensor notation)}$$

Equilibrium Equations:

The stresses and body force vector $\boldsymbol{f}$ at each point satisfy the following three equilibrium equations for elastostatic problems:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + f_x = 0,$$
$$\frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + f_y = 0, \tag{6.6}$$
$$\frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + f_z = 0.$$

Or in index or tensor notation:

$$\sigma_{ij,j} + f_i = 0.$$

Boundary Conditions (BCs):

At each point on the boundary Γ and in each direction, either displacement or traction (stress on the boundary) should be given, that is:

$$u_i = \bar{u}_i, \qquad on\ \ \Gamma_u\ (specified\ \ displacement);$$
$$t_i = \bar{t}_i, \qquad on\ \ \Gamma_\sigma\ (specified\ \ traction); \tag{6.7}$$

in which the barred quantities denote given values, and the traction (stress on a surface) is defined by $t_i = \sigma_{ij} n_j$, or in a matrix form:

$$\begin{Bmatrix} t_x \\ t_y \\ t_z \end{Bmatrix} = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix} \begin{Bmatrix} n_x \\ n_y \\ n_z \end{Bmatrix}.$$

with *n* being the normal (Figure 6.2).

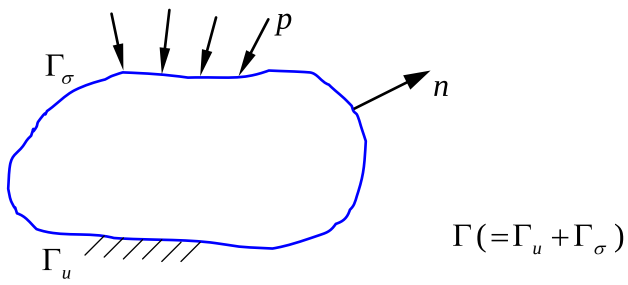


Figure 6.2. The boundary of an elastic domain.

Stress Analysis:

For 3-D stress analysis, one needs to solve equations in (6.3), (6.5) and (6.6) under the BCs in (6.7) in order to obtain the stress, strain and displacement fields (15 equations for 15 unknowns for 3-D problems). Analytical solutions are often difficult to find and thus numerical methods such as the FEA is applied in 3-D stress analysis.

II. Finite Element Formulation

We first summarize the FEA formulation for 3-D elasticity problems, which are straight forward extensions of the FEA formulations for 1-D bar and 2-D elasticity problems.

Displacement Field:

As in the FEA formulations for 1-D and 2-D problems, we first interpolate the displacement fields within a 3-D element using shape functions N_i:

$$\begin{aligned} u &= \sum_{i=1}^{N} N_i u_i, \\ v &= \sum_{i=1}^{N} N_i v_i, \\ w &= \sum_{i=1}^{N} N_i w_i, \end{aligned} \tag{6.8}$$

in which u_i, v_i, and w_i are nodal values of the displacement on the element, and N is the number of nodes on that element. In matrix form, we have:

$$
\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_{(3\times1)} = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & \cdots \\ 0 & N_1 & 0 & 0 & N_2 & 0 & \cdots \\ 0 & 0 & N_1 & 0 & 0 & N_2 & \cdots \end{bmatrix}_{(3\times3N)} \begin{Bmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ v_2 \\ w_2 \\ \vdots \end{Bmatrix}_{(3N\times1)} \tag{6.9}
$$

Or in a matrix form:

$$\mathbf{u} = \mathbf{N}\mathbf{d}.$$

Using relations (6.5) and (6.8), we can derive the strain vector to obtain:

$$\boldsymbol{\varepsilon} = \mathbf{B}\,\mathbf{d}$$

in which **B** is the matrix relating the nodal displacement vector **d** to the strain vector **ε**. Note that the dimensions of the **B** matrix are $6\times3N$.

Stiffness Matrix:

Once the **B** matrix is found, one can apply the following familiar expression to determine the stiffness matrix for the element:

$$\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E} \mathbf{B}\, dv. \tag{6.10}$$

The dimensions of the stiffness matrix **k** are $3N\times3N$. A numerical quadrature is often needed to evaluate the above integration, which can be expensive if the number of nodes is large, such as for higher-order elements.

A Note of the Rigid-Body Motions:

Note that there are six rigid-body motions for 3-D bodies:

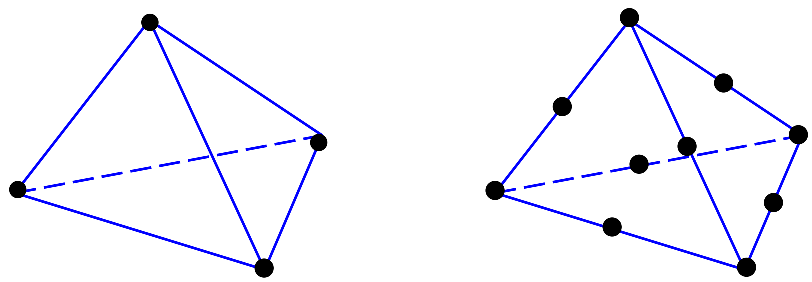
3 translations and 3 rotations.

These rigid-body motions (causes of singularity of the system of equations) must be removed from the FEA model for stress analysis to ensure the accuracy of the analysis. On the other hand, over constrains can also cause inaccurate or unwarranted results.

III. Typical 3-D Solid Elements

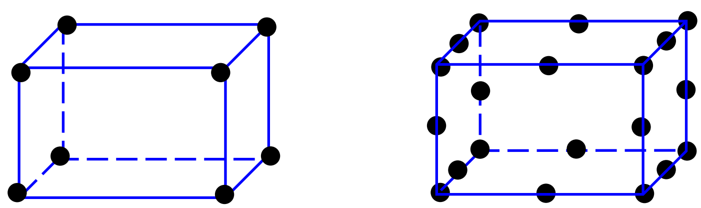
We can classify the type of elements for 3-D problems as follows (Figure 6.3)according to their shapes and the orders of the shape functions constructed on the elements:

Tetrahedron:



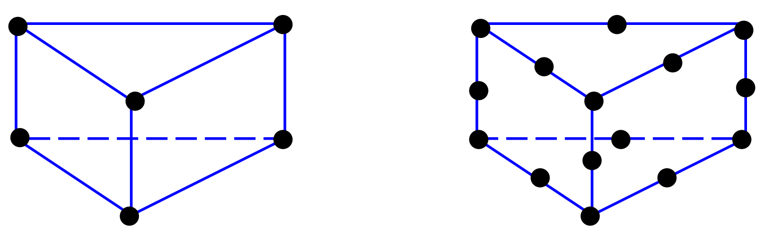
linear (4 nodes) *quadratic (10 nodes)*

Hexahedron (brick):



linear (8 nodes) *quadratic (20 nodes)*

Penta:



linear (6 nodes) *quadratic (15 nodes)*

Figure 6.3. Different types of 3-D solid elements.

Whenever possible, try to apply higher-order (quadratic) elements, such as 10-node tetrahedron and 20-node brick elements for 3-D stress analysis. Avoid using the linear, especially the 4-node tetrahedron elements in 3-D stress analysis, because they are inaccurate for such purposes. However, it is fine to use them for deformation analysis or in vibration analysis (see next chapter).

In the following section, we will examine the element formulation for the 8-node brick element.

Element Formulation:

Linear Hexahedron Element

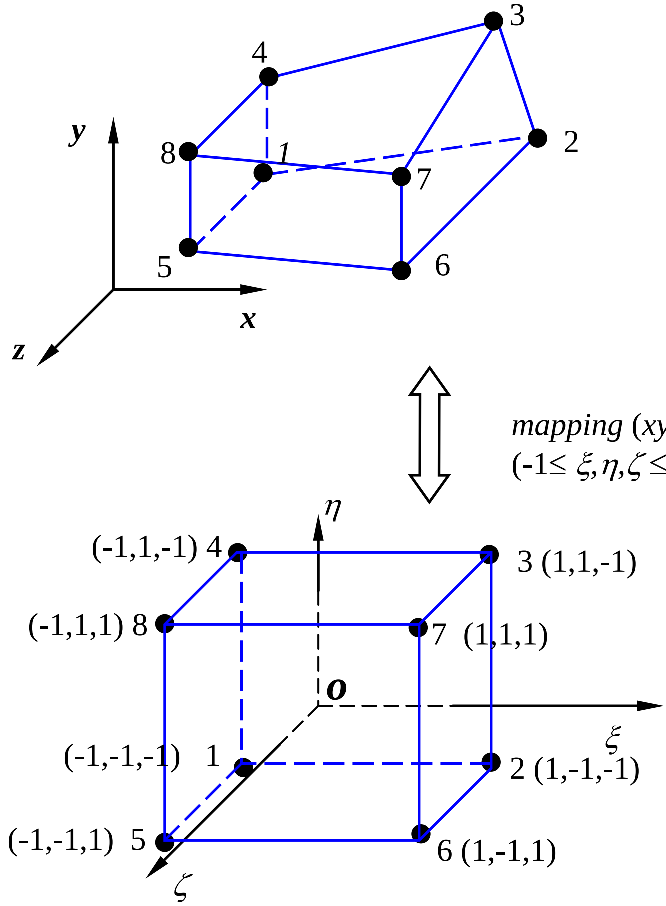


Figure 6.4. Mapping an element to the natural coordinate system.

Displacement Field in the Element:

$$u = \sum_{i=1}^{8} N_i u_i, \qquad v = \sum_{i=1}^{8} N_i v_i, \qquad w = \sum_{i=1}^{8} N_i w_i. \tag{6.11}$$

Shape Functions:

$$
\begin{aligned}
N_1(\xi,\eta,\zeta) &= \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta) \quad , \\
N_2(\xi,\eta,\zeta) &= \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta) \quad , \\
N_3(\xi,\eta,\zeta) &= \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta) \quad , \\
&\ \ \vdots \qquad\qquad \vdots \\
N_8(\xi,\eta,\zeta) &= \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta) \quad .
\end{aligned}
\tag{6.12}
$$

Note that we have the following relations for the shape functions:

$$
\begin{aligned}
&N_i(\xi_j,\eta_j,\zeta_j) = \delta_{ij} \ , \quad i, j = 1,2,\cdots,8. \\
&\sum_{i=1}^{8} N_i(\xi,\eta,\zeta) = 1.
\end{aligned}
$$

Coordinate Transformation (Mapping):

$$
x = \sum_{i=1}^{8} N_i\, x_i, \qquad y = \sum_{i=1}^{8} N_i\, y_i, \qquad z = \sum_{i=1}^{8} N_i\, z_i. \tag{6.13}
$$

That is, the same shape functions are used as for the displacement field. This is again called an isoparametric element.

Jacobian Matrix:

$$
\begin{Bmatrix} \dfrac{\partial u}{\partial \xi} \\ \dfrac{\partial u}{\partial \eta} \\ \dfrac{\partial u}{\partial \zeta} \end{Bmatrix}
= \underbrace{\begin{bmatrix} \dfrac{\partial x}{\partial \xi} & \dfrac{\partial y}{\partial \xi} & \dfrac{\partial z}{\partial \xi} \\ \dfrac{\partial x}{\partial \eta} & \dfrac{\partial y}{\partial \eta} & \dfrac{\partial z}{\partial \eta} \\ \dfrac{\partial x}{\partial \zeta} & \dfrac{\partial y}{\partial \zeta} & \dfrac{\partial z}{\partial \zeta} \end{bmatrix}}_{\equiv\ \mathbf{J}\quad \text{Jacobian matrix}}
\begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \\ \dfrac{\partial u}{\partial z} \end{Bmatrix}. \tag{6.14}
$$

Inverting this relation, we have:

$$
\begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \\ \dfrac{\partial u}{\partial z} \end{Bmatrix}
= \mathbf{J}^{-1}
\begin{Bmatrix} \dfrac{\partial u}{\partial \xi} \\ \dfrac{\partial u}{\partial \eta} \\ \dfrac{\partial u}{\partial \zeta} \end{Bmatrix}, \quad \text{with } \frac{\partial u}{\partial \xi} = \sum_{i=1}^{8} \frac{\partial N_i}{\partial \xi} u_i \text{ and so on.} \tag{6.15}
$$

and similarly for *v* and *w*. These relations lead to the following expression for the strain:

$$\boldsymbol{\varepsilon} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{\partial w}{\partial z} \\ \dfrac{\partial v}{\partial x} + \dfrac{\partial u}{\partial y} \\ \dfrac{\partial w}{\partial y} + \dfrac{\partial v}{\partial z} \\ \dfrac{\partial u}{\partial z} + \dfrac{\partial w}{\partial x} \end{Bmatrix} = \cdots_{use\,(6.15)} = \mathbf{B}\,\mathbf{d}\,.$$

where **d** is the nodal displacement vector, that is:

$$\boldsymbol{\varepsilon} = \mathbf{B}\,\mathbf{d}. \tag{6.16}$$

Strain energy is evaluated as:

$$\begin{aligned} U &= \frac{1}{2}\int_V \boldsymbol{\sigma}^T\, \boldsymbol{\varepsilon}\, dV = \frac{1}{2}\int_V (\mathbf{E}\boldsymbol{\varepsilon})^T\, \boldsymbol{\varepsilon}\, dV \\ &= \frac{1}{2}\int_V \boldsymbol{\varepsilon}^T \mathbf{E}\, \boldsymbol{\varepsilon}\, dV \\ &= \frac{1}{2}\, \mathbf{d}^T \left[\int_V \mathbf{B}^T \mathbf{E}\, \mathbf{B}\, dV \right] \mathbf{d}. \end{aligned} \tag{6.17}$$

That is, the element stiffness matrix is

$$\mathbf{k} = \int_V \mathbf{B}^T \mathbf{E}\, \mathbf{B}\; dV. \tag{6.18}$$

In $\xi\eta\zeta$ coordinates:

$$dV = (\det \mathbf{J})\, d\xi\, d\eta\, d\zeta \tag{6.19}$$

Therefore,

$$\mathbf{k} = \int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1} \mathbf{B}^T \mathbf{E}\, \mathbf{B}\, (\det \mathbf{J})\, d\xi\, d\eta\, d\zeta\,. \tag{6.20}$$

It is easy to verify that the dimensions of this stiffness matrix is 24x24.

Note that in general, 3-D elements do not use rotational DOFs.

Treatment of Distributed Loads:

Distributed loads need to be converted into nodal forces using the equivalent energy concept as discussed in earlier chapters. Figure 6.5 shows the result of a pressure load converted to nodal forces for a 20-node hexahedron element. Note the direction of the forces at the four corner nodes, which is not intuitive at all.

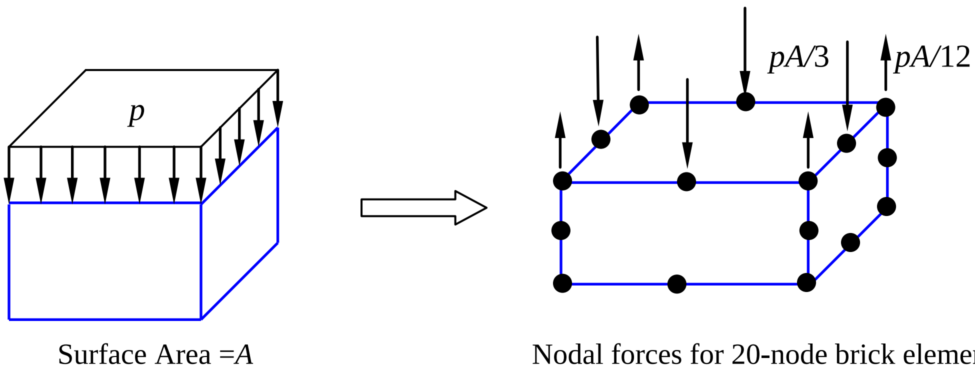


Figure 6.5. Equivalent nodal forces on a 20-node brick element for a constant distributed load *p*.

Stresses:

To compute the the stresses within an element, one uses the following relation once the nodal displacement vector is known for that element:

$$\boldsymbol{\sigma} = \mathbf{E}\boldsymbol{\varepsilon} = \mathbf{EBd}.$$

Stresses are evaluated at selected points (Gaussian points or nodes) on each element. Stress values at the nodes are often discontinuous and less accurate. Averaging of the stresses from surrounding elements around a node are often employed to smooth the stress field results.

The von Mises stress for 3-D problems is given by:

$$\sigma_e = \sigma_{VM} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}\,. \tag{6.21}$$

where σ_1, σ_2 and σ_3 are the three principal stresses.

3-D stress analysis using solid elements is one of the most difficult tasks in FEA. Meshing structures with complicated geometries can be very tedious and time consuming. Great care need to be taken to make sure that the FEA mesh is in good quality (for example, with no distorted elements). Computing cost is another factor. For structures with stress concentrations, large FEA models are often needed, which can run hours or days to solve even on today's best computers. A good CAE engineer should be able to decided where to apply a fine mesh and where not to, in order to trike a balance between the cost and accuracy for an FEA task.

Examples:

Figure 6.6 show a drag link FEA model using solid elements. Although the structure has a slender shape, it has a bended angle and holes. 3-D solid elements are needed for the stress analysis in this case. Great care is taken in meshing this part, where 20-node brick elements are used for better accuracy in the stress analysis. Buckling analysis may also be conducted for slender structures when they are under compressions. More information about buckling analysis using FEA can be found in the references or in documents of FEA software packages.

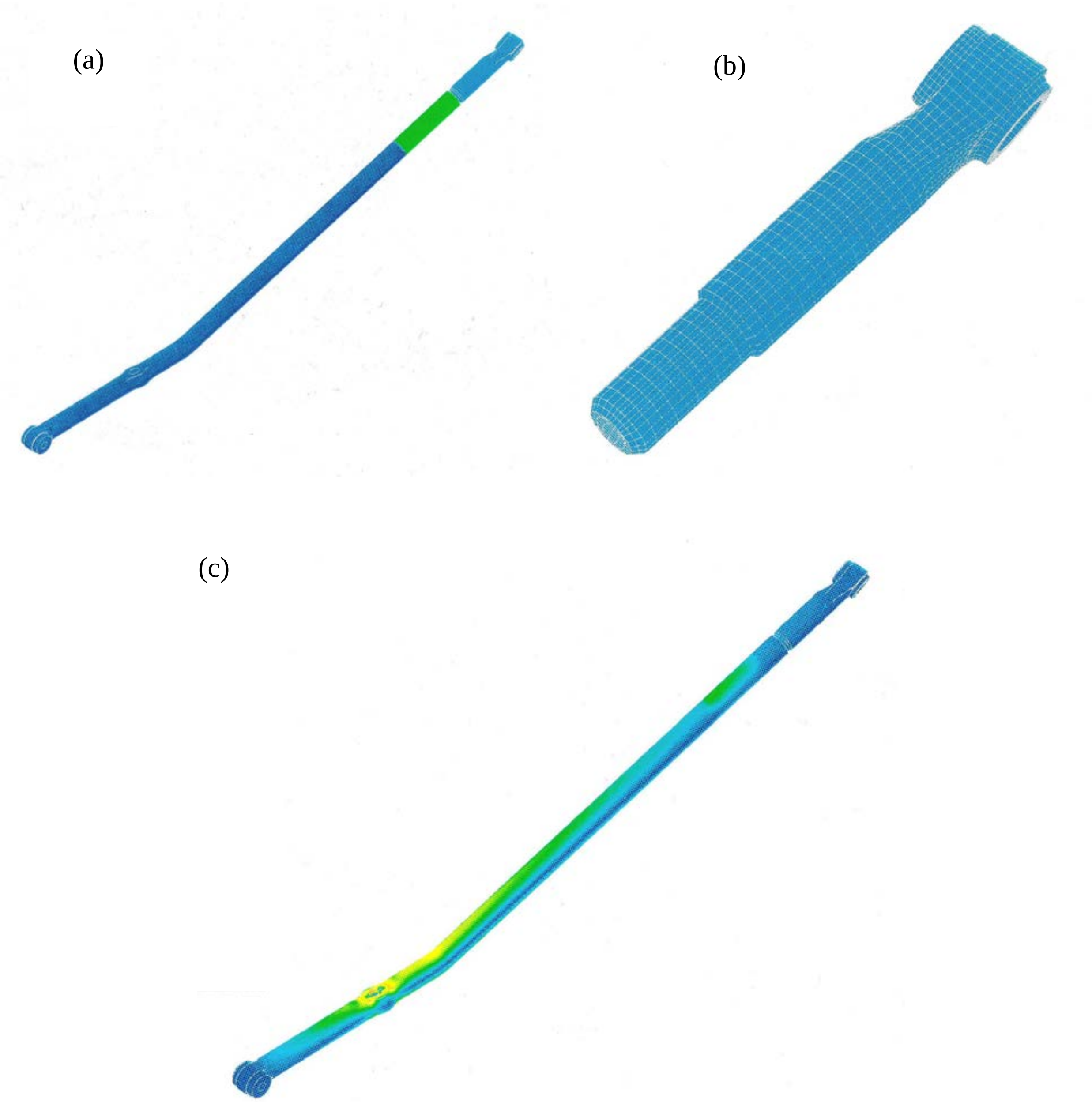
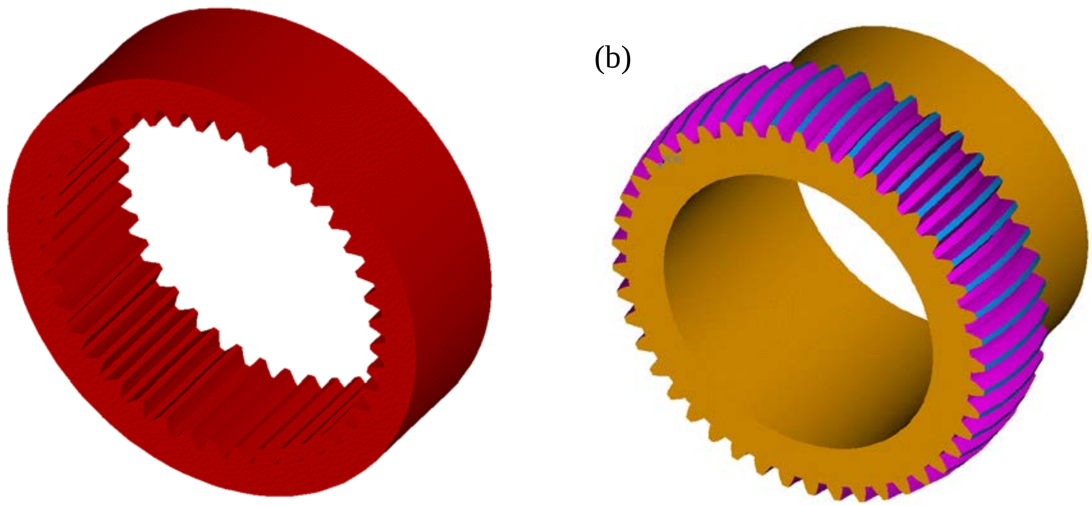


Figure 6.6. FEA for a drag link: (a) The model; (b) Mesh for the right end; (c) Stress distribution due to tension loads applied at the two ends.

Figure 6.7 show a 3-D FEA of a gear coupling which is applied to transmit powers through two aligned rotating shafts. Contact stresses and failure modes are to be determined based on detailed 3-D FE models. This analysis requires the use of nonlinear FEA options, which are readily available now in almost all FEA software packages.

(a)

(b)



(c)

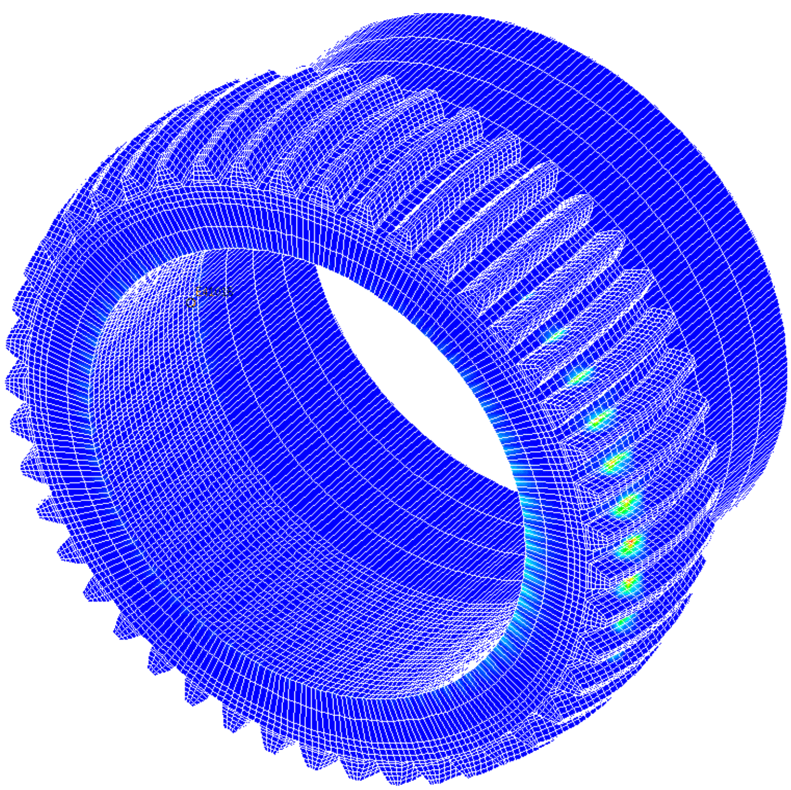
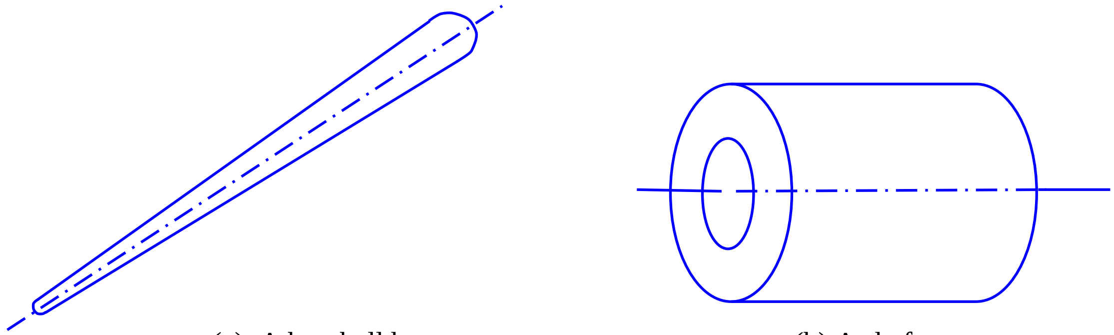


Figure 6.7. Analysis of a gear coupling: (a) Ring gear; (b) Hub gear; (c) High contact stresses in the gear teeth obtained using nonlinear FEA.

IV. Solids of Revolution (Axisymmetric Analysis)

Many mechanical parts can be considered as solids of revolution or axisymmetric solids (Figure 6.8). If the applied loads on these axisymmetric parts are also axisymmetric, the 3-D analysis can be simplified by using the axisymmetric models.



(a) A baseball bat (b) A shaft

Figure 6.8. Examples of axisymmetric solids.

Cylindrical Coordinates:

Axisymmetric models are based on the cylindrical coordinate (r, θ, z) (Figure 6.9).

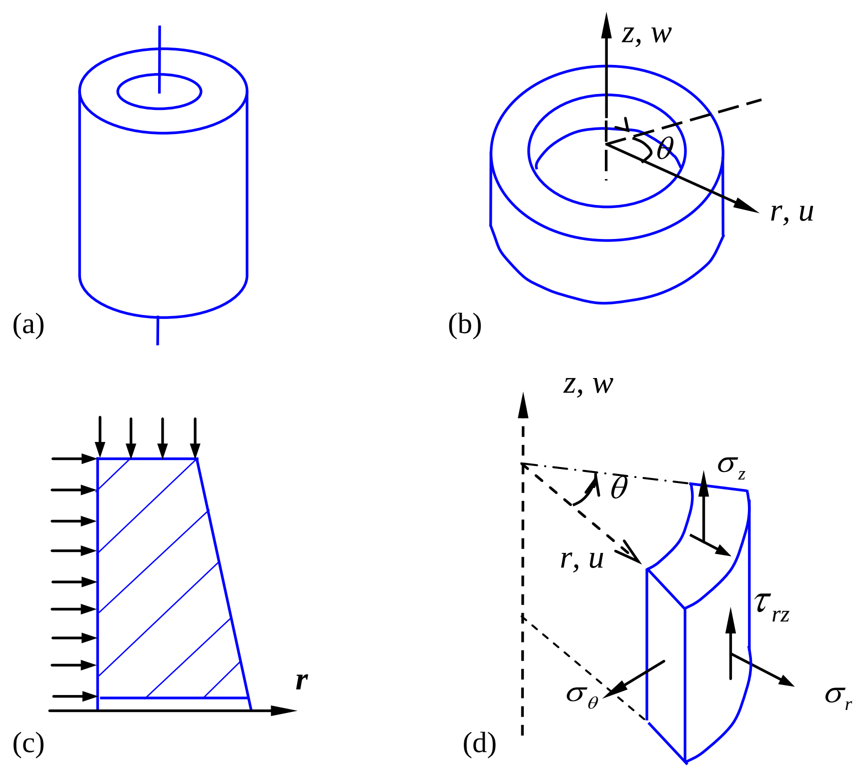


Figure 6.9. (a) An axisymmetric body; (b) The cylindrical coordinates; (c) An axisymmetric model; (d) Stress components in the cylindrical coordinates.

In this cylindrical coordinates, one can establish the strain-displacement and stress-strain relations, and the equilibrium equations.

Displacement Field:

$$u = u(r,z), \quad w = w(r,z), \quad v = 0 \quad (\text{No circumferential component}).$$

Strain-Displacement Relation* (Figure 6.10)*:

$$\begin{aligned} &\varepsilon_r = \frac{\partial u}{\partial r}, \qquad \varepsilon_\theta = \frac{u}{r}, \qquad \varepsilon_z = \frac{\partial w}{\partial z}, \\ &\gamma_{rz} = \frac{\partial w}{\partial r} + \frac{\partial u}{\partial z}, \qquad \gamma_{r\theta} = \gamma_{z\theta} = 0. \end{aligned} \tag{6.22}$$

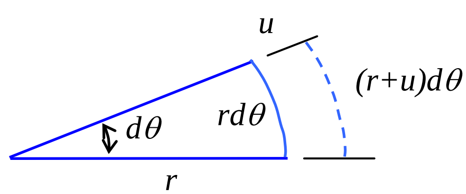


Figure 6.10. The geometric relations used in deriving strain-displacement relations.

Stresses-Strain Relation:

$$\begin{Bmatrix} \sigma_r \\ \sigma_\theta \\ \sigma_z \\ \tau_{rz} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_r \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{rz} \end{Bmatrix} \tag{6.23}$$

Axisymmetric Elements* (Figure 6.11)*:

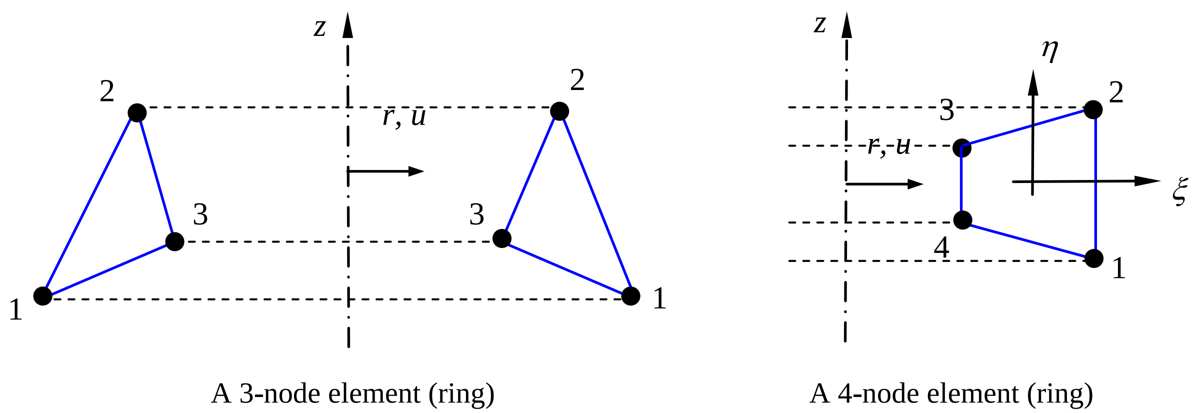


Figure 6.11. Axisymmetric 3-node and 4-node elements.

Formulation of the axisymmetric elements are similar to other 2-D plane stress/strain elements. The stiffness matrix, for example, is given by:

$$\mathbf{k} = \int_V \mathbf{B}^T \, \mathbf{E} \, \mathbf{B} \, r dr \, d\theta \, dz \,, \tag{6.24}$$

or

$$\begin{aligned}\mathbf{k} &= \int_0^{2\pi} \int_{-1}^{1} \int_{-1}^{1} \mathbf{B}^T \mathbf{E} \, \mathbf{B} \, r (\det \mathbf{J}) \, d\xi \, d\eta \, d\theta \\ &= 2\pi \int_{-1}^{1} \int_{-1}^{1} \mathbf{B}^T \mathbf{E} \, \mathbf{B} \, r (\det \mathbf{J}) \, d\xi \, d\eta,\end{aligned} \tag{6.25}$$

in which **B** is the matrix relating the nodal displacement vector to strain vector in the cylindrical coordinate system.

These axisymmetric elements have planar shapes, but actually represent rings in the circumferential directions of the axisymmetric solids.

Applications:

Many rotating parts, such as a flywheel (Figure 6.12), can be modeled using axisymmetric elements.

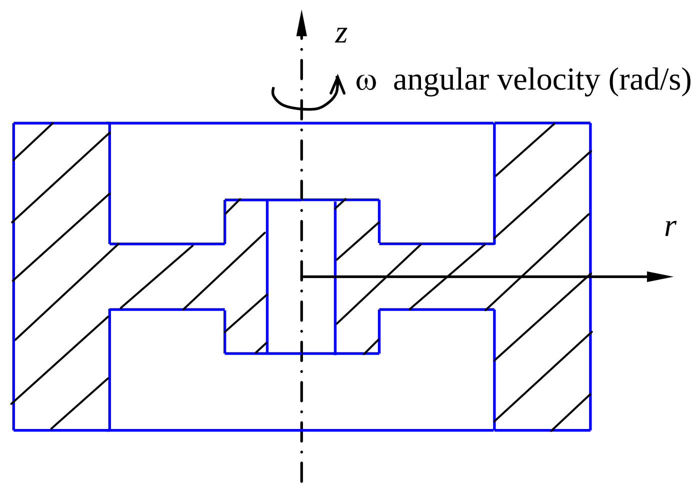


Figure 6.12. Cross-section of a flywheel that is rotating with an angular velocity ω.

Body forces in rotating parts are given by the following formulas:

$$\begin{aligned} f_r &= \rho \, r \, \omega^2, && \text{equivalent radial centrifugal/inertial force;} \\ f_z &= -\rho \, g, && \text{gravitational force;} \end{aligned} \tag{6.26}$$

where ρ is the mass density and g is gravitational acceleration.

Other examples include cylinders subject to internal pressure (Figure 6.13) and press fit (Figure 6.14).

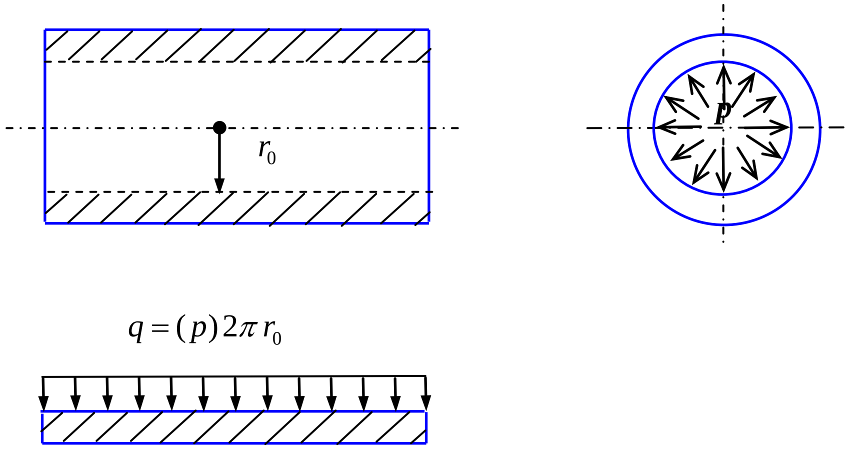
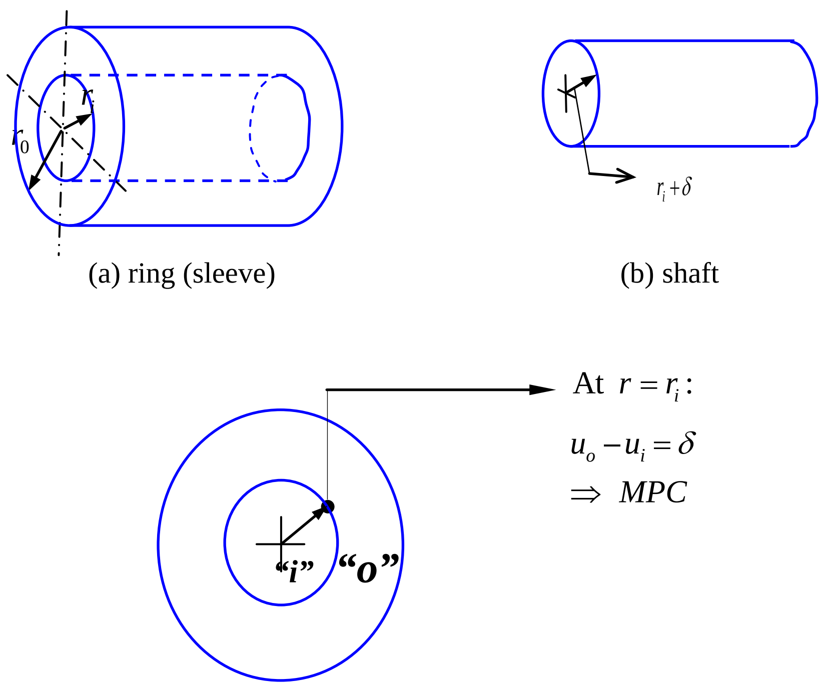


Figure 6.13. A cylinder with internal radius r_0 is subjected to internal pressure p.



(a) ring (sleeve)

(b) shaft

(c) Interface condition (Multi-Point Constraint (MPC))

Figure 6.14. An example of press fit.

A more advanced example is the Belleville (conical) spring shown in Figure 6.15:

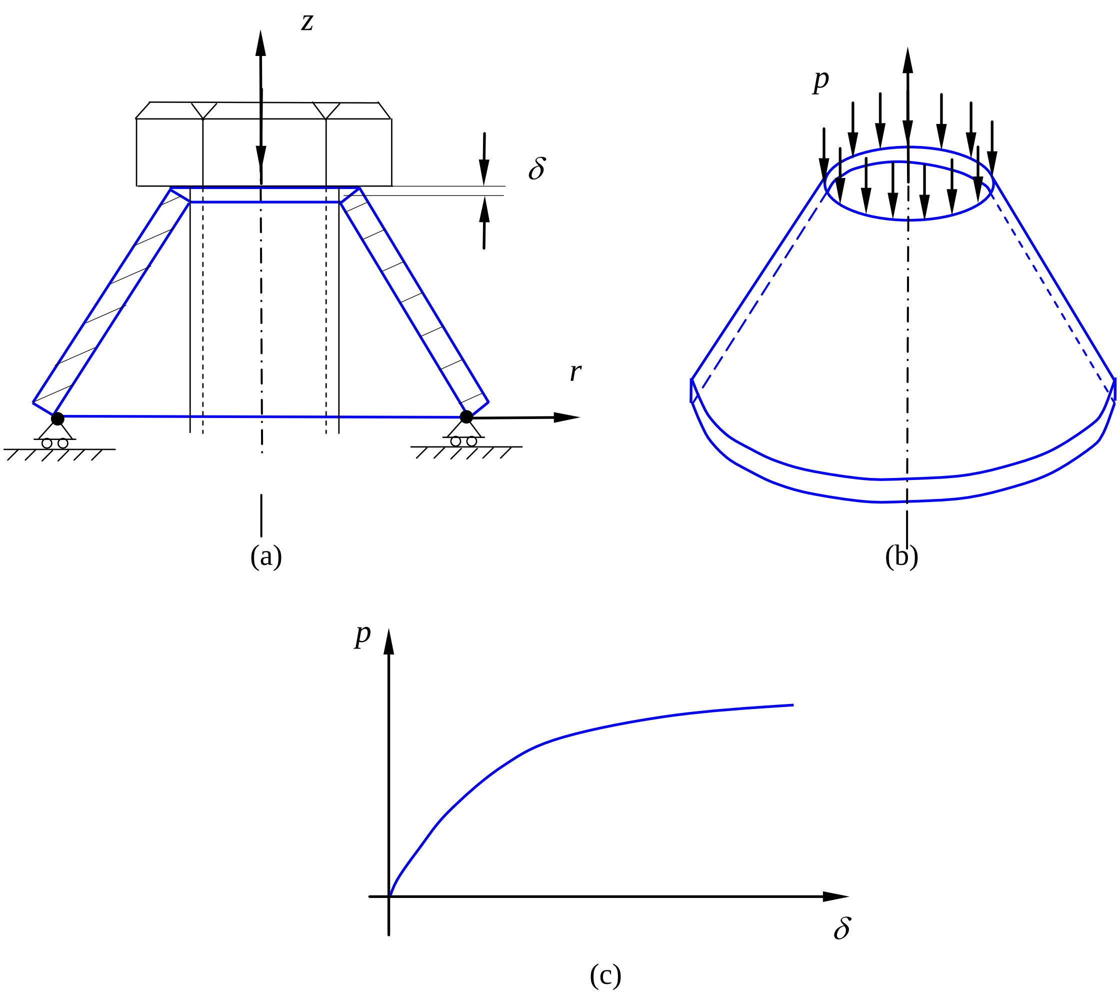


Figure 6.15. A Belleville (conical) springs.

This is a geometrically nonlinear (large deformation) problem (because of the nonlinear behavior shown in the force-displacement curve as in Figure 6.15(c). Iteration approaches (incremental methods) need to be employed to solve this type of problems.

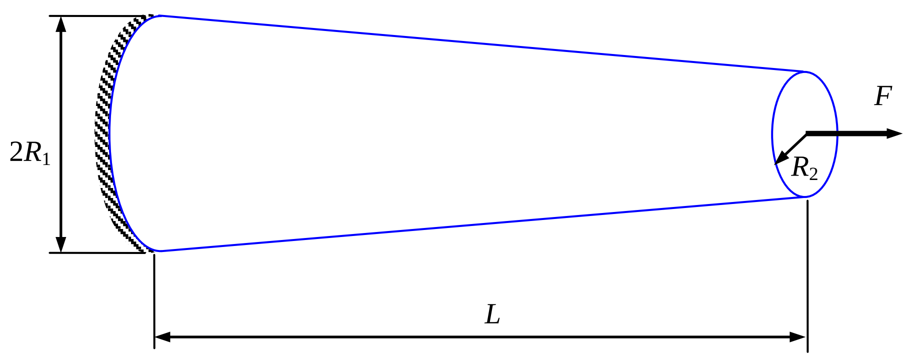
Examples of axisymmetric solids shown in Figures 6.12-15 can be used as test problems. One can build both 3-D solid models and corresponding 2-D axisymmetric models to compare the modeling and computational efficiencies. The accuracy of the results for the two type of models should be within certain tolerance if correct boundary conditions and same mesh patterns/densities are employed.

V. Summary

In this chapter, we discussed the 3-D (solid) elements for elasticity problems, that is, general 3-D deformation and stress analyses. Solid elements are the most accurate elements and should be applied when the bar, beam, plane stress/strain, plate/shell elements are no longer valid or accurate. Especially for stress concentration problems, higher-order solid elements, such as 10-node tetrahedron or 20-node hexahedron (brick) elements should be employed in the FEA. For rotating parts or solids of revolution and under axisymmetric loading, the axisymmetric elements are most effective and efficient.

VI. Problems

Problem 1. For a tapered bar shown below, study the deformation and stresses in the bar with a 3-D model using solid elements and a 1-D model using 1-D bar elements. Assume R_1 = 1 m, R_2 = 0.5 m, L = 5 m, force F = 3000 N, the Young's modulus E = 200 GPa and Poisson's ratio υ = 0.3. The bar is fixed at the left end. Compare the results from the 3-D model and 1-D model.

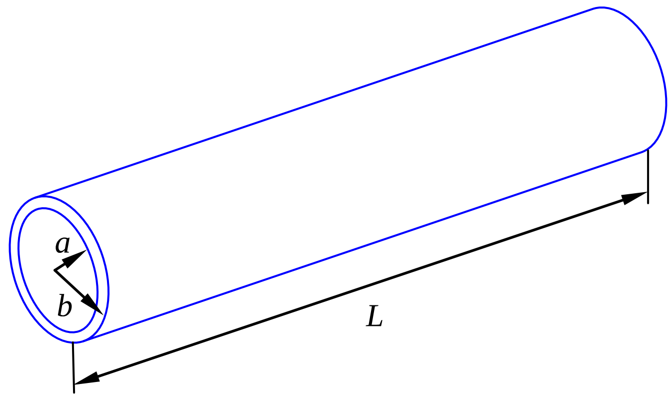


Problem 2. An open cylinder shown below has an inner radius a = 1 m, outer radius b = 1.1 m, length L = 10 m and is applied with a pressure load p = 10 GPa on the inner surface.

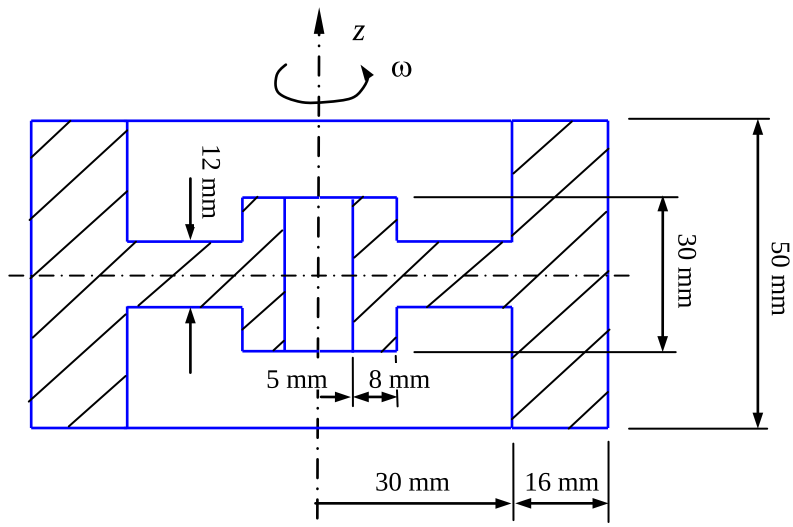
(a) Compute the stresses in the cylinder using 2-D plane stress, 2-D axisymmetric and 3-D solid models. Compare the FEA results based on these models.

(b) Consider the same cylinder, added with a hole of radius r = 0.2 m on the cylinder wall, centered at the mid-length, and along the radial direction.

Assume the Young's modulus E = 200 GPa and Poisson's ratio υ = 0.3.



Problem 3. For the rotating part sketched below, assume that it is made of steel with the Young's modulus E = 200 GPa, Poisson's ratio υ = 0.3, and mass density ρ = 7850 kg/m^3. Assume that the part is rotating at a speed of 1000 RPM about the z axis. Ignore the gravitational force. Compute the stresses in the part using the FEA with a full 3-D model and an axisymmetric model. Compare the results with the two models.



Chapter 7. Structural Vibration and Dynamics

In this chapter, we first review the basic equations and their solutions for structural vibration and dynamic analysis. Then, we discuss the FEA formulations for solving vibration and dynamic responses. Guidelines in modeling and solving such problems are provided.

There are three main types of problems for structural vibration and dynamic analyses:

- Natural frequencies and modes ($f(t) = 0$);
- Frequency response ($f(t) = f_o \sin\omega t$);
- Transient response ($f(t)$ is arbitrary);

where $f(t)$ is the dynamic force applied on the structure, t the time, and ω the circular frequency (Figure 7.1)

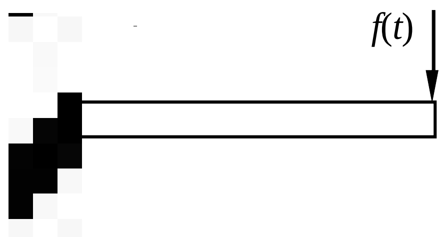


Figure 7.1. A dynamic force applied to the structure.

I. Basic Equations

A. Single DOF System

First, let us consider a single degree of freedom (DOF) system shown in Figure 7.2.

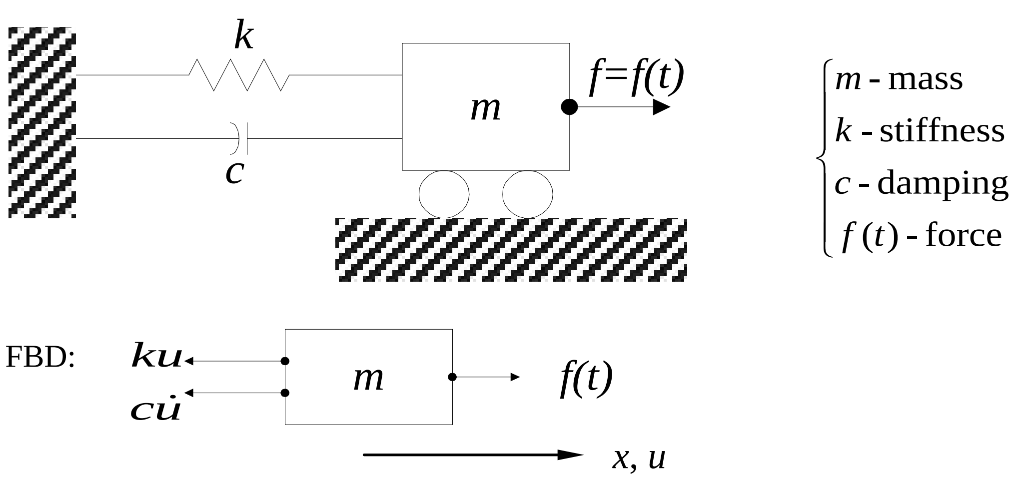


Figure 7.2. A single DOF system with damping.

From the free-body diagram (FBD) and Newton's law of motion ($ma = f$), we have:

$$m\ddot{u} = f(t) - ku - c\dot{u} \, ,$$

that is:

$$m\ddot{u} + c\dot{u} + ku = f(t) \, , \tag{7.1}$$

where u is the displacement, $\dot{u} = du/dt$ the velocity, and $\ddot{u} = d^2u/dt^2$ the acceleration.

Free Vibration (Normal Mode Analysis):

If the applied force $f(t) = 0$ and no damping ($c = 0$), Eq. (7.1) becomes:

$$m\ddot{u} + ku = 0 \, . \tag{7.2}$$

The physical meaning of this equation is: inertia force + elastic/stiffness force = 0.

Although there is no applied force, the structural can have nonzero displacement or experience vibrations under contain initial conditions (ICs). To solve for such nontrivial solutions, we assume:

$$u(t) = U \sin \omega t \, ,$$

where ω is the circular frequency of oscillation, U the amplitude. Substituting this into Eq. (7.2) yields:

$$-U\omega^2 m \sin \omega t + kU \sin \omega t = 0$$

that is:

$$\left[-\omega^2 m + k \right] U = 0 \, .$$

For nontrivial solutions for U, we must have:

$$\left[-\omega^2 m + k \right] = 0 \, ,$$

which yields

$$\omega = \sqrt{\frac{k}{m}} \, . \tag{7.3}$$

This is the circular *natural frequency* of the single DOF system (rad/s). The cyclic frequency ($1/s$ = Hz) is $\omega/2\pi$.

Equation (7.3) is a very important result in free vibration analysis, which says that the natural frequency of a structural is proportional to the square-root of the stiffness of the structure and inversely proportional to the square-root of the total mass of the structure.

The typical response of the system in undamped free vibration is sketched in Figure 7.3.

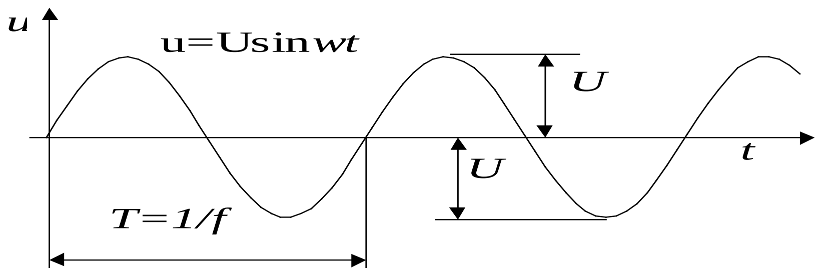


Figure 7.3. Typical response in an undamped free vibration.

For non-zero damping c, where

$$0 < c < c_c = 2m\omega = 2\sqrt{k\,m} \quad (c_c = \text{critical damping}) \tag{7.4}$$

we have the *damped natural frequency*:

$$\omega_d = \omega\sqrt{1-\xi^2}\,, \tag{7.5}$$

where

$$\xi = c / c_c \tag{7.6}$$

is called the damping ratio.

For structural damping: $0 \le \xi < 0.15$ (usually 1~5%)

$$\omega_d \approx \omega\,. \tag{7.7}$$

That is, we can ignore damping in normal mode analysis.

The typical response of a system in damped free vibration is sketched in Figure 7.4. We can see that damping has the effect of reducing the vibration of the system.

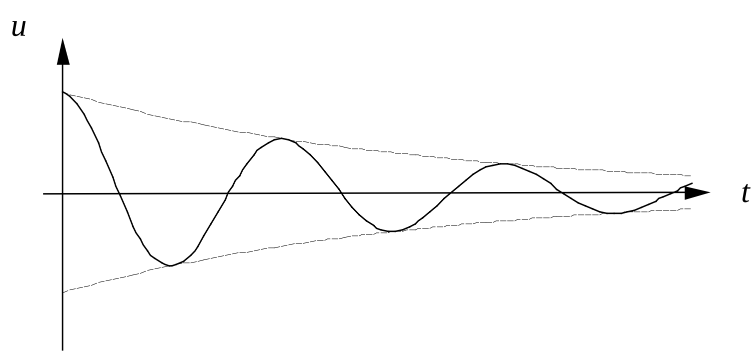


Figure 7.4. Typical response of a free vibration with a nonzero damping $c < c_c$.

B. Multiple DOF Systems

Equation of Motion:

For multiple DOF systems, the equation of motion can be written as:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{f}(t) , \tag{7.8}$$

in which: $\mathbf{u}$ — nodal displacement vector;

$\mathbf{M}$ — mass matrix of the structure;

$\mathbf{C}$ — damping matrix;

$\mathbf{K}$ — stiffness matrix;

$\mathbf{f}$ — forcing vector.

The physical meaning of Eq. (7.8) is :

Inertia forces + Damping forces + Elastic forces = Applied forces

We already know how to determine the stiffness matrix $\mathbf{K}$ for a structure, as discussed in previous chapters. The main tasks in vibration analysis is to determine the mass matrix and damping matrix for the structure.

Mass Matrices:

There are two types of mass matrices: *lumped mass matrices* and *consistent mass matrices*. The former is empirical and easier to determine, and the latter is analytical and more involved in their computing.

We use a bar element to illustrate the lumped mass matrix (Figure 7.5).

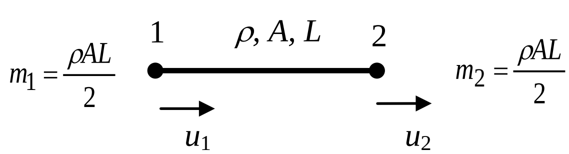


Figure 7.5. The lumped mass for a 1-D bar element.

For this bar element, the lumped mass matrix for the element is found to be:

$$\mathbf{m} = \begin{bmatrix} \dfrac{\rho AL}{2} & 0 \\ 0 & \dfrac{\rho AL}{2} \end{bmatrix},$$

which is a diagonal matrix and thus is easier to compute.

In general, we apply the following element *consistent mass matrix*:

$$\mathbf{m} = \int_V \rho \mathbf{N}^T \mathbf{N} dV \ , \tag{7.9}$$

where **N** is the same shape function matrix as used for the displacement field, and V is the volume of the element.

Equation (7.9) is obtained by considering the kinetic energy within an element:

$$\begin{aligned}
\mathrm{K} &= \frac{1}{2}\dot{\mathbf{u}}^T \mathbf{m}\dot{\mathbf{u}} \qquad\qquad (\text{cf. } \frac{1}{2}mv^2) \\
&= \frac{1}{2}\int_V \rho \dot{u}^2 dV \ = \frac{1}{2}\int_V \rho\left(\dot{u}\right)^T \dot{u} dV \\
&= \frac{1}{2}\int_V \rho\left(\mathbf{N}\dot{\mathbf{u}}\right)^T \left(\mathbf{N}\dot{\mathbf{u}}\right) dV \\
&= \frac{1}{2}\dot{\mathbf{u}}^T \underbrace{\int_V \rho \mathbf{N}^T \mathbf{N} dV}_{\mathbf{m}} \ \dot{\mathbf{u}}.
\end{aligned} \tag{7.10}$$

For the bar element (linear shape function), the consistent mass matrix is:

$$\begin{aligned}
\mathbf{m} &= \int_V \rho \begin{bmatrix} 1-\xi \\ \xi \end{bmatrix} \begin{bmatrix} 1-\xi & \xi \end{bmatrix} AL d\xi \\
&= \rho AL \begin{bmatrix} 1/3 & 1/6 \\ 1/6 & 1/3 \end{bmatrix} \begin{matrix} \ddot{u}_1 \\ \ddot{u}_2 \end{matrix}
\end{aligned} \tag{7.11}$$

which is a non-diagonal matrix.

As in the case for stiffness matrices, element mass matrices are established in local coordinates first, then transformed to global coordinates, and finally assembled together to form the global structure mass matrix **M**.

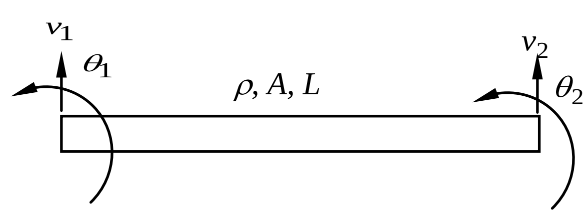


Figure 7.6. The lumped mass for a 1-D simple beam element.

For a simple beam element (Figure 7.6), the consistent mass matrix can be found readily by applying the four shape functions listed in Eq. (2.41). We have:

$$
\begin{aligned}
\mathbf{m} &= \int_{\mathrm{V}} \rho \mathbf{N}^T \mathbf{N} dV \\
&= \frac{\rho A L}{420} \begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix} \begin{array}{c} \ddot{v}_1 \\ \ddot{\theta}_1 \\ \ddot{v}_2 \\ \ddot{\theta}_2 \end{array}
\end{aligned} \tag{7.12}
$$

Selecting a proper unit system is very important in vibration or dynamic analysis. Two choices of the units are listed in Table 7.1. Make sure they are consistent in the FEA models.

Table 7.1. Units in dynamic analysis

	Choice I	*Choice II*
t (time)	s	s
L (length)	m	mm
m (mass)	kg	Mg
a (accel.)	m/s^2	mm/s^2
f (force)	N	N
ρ *(density)*	kg/m^3	Mg/mm^3

II. Free Vibration of Multiple DOF Systems

Free vibration or normal mode analysis aims to study the dynamic characteristics of a structure, which include:

- Natural frequencies;
- Normal modes (shapes).

Let $\mathbf{f}(t) = \mathbf{0}$ and $\mathbf{C} = \mathbf{0}$ (ignore damping) in the dynamic equation (7.8) and obtain:

$$\mathbf{M\ddot{u}} + \mathbf{Ku} = \mathbf{0} \tag{7.13}$$

Assume that displacements vary harmonically with time, that is:

$$
\begin{aligned}
\mathbf{u}(t) &= \overline{\mathbf{u}} \sin(\omega t), \\
\dot{\mathbf{u}}(t) &= \omega \overline{\mathbf{u}} \cos(\omega t), \\
\ddot{\mathbf{u}}(t) &= -\omega^2 \overline{\mathbf{u}} \sin(\omega t),
\end{aligned}
$$

where $\overline{\mathbf{u}}$ is the vector of the amplitudes of the nodal displacements.

Substitutes these into Eq. (7.13) yields:

$$\left[\mathbf{K} - \omega^2 \mathbf{M}\right] \overline{\mathbf{u}} = \mathbf{0}. \tag{7.14}$$

This is a generalized eigenvalue problem (EVP). The trivial solution is $\overline{\mathbf{u}} = \mathbf{0}$ for any values of ω (not interesting). Nontrivial solutions ($\overline{\mathbf{u}} \neq \mathbf{0}$) exist if and only if:

$$\left|\mathbf{K} - \omega^2 \mathbf{M}\right| = 0 \tag{7.15}$$

This is an n-th order polynomial of ω^2, from which we can find n solutions (roots) or eigenvalues ω_i (i = 1, 2, …, n). These are the natural frequencies (or characteristic frequencies) of the structure.

The smallest non-zero eigenvalue ω_1 is called the *fundamental frequency*.

For each ω_i , Eq. (7.14) gives one solution or eigen vector:

$$\left[\mathbf{K} - \omega_i{}^2 \mathbf{M}\right] \overline{\mathbf{u}}_i = \mathbf{0}.$$

$\overline{\mathbf{u}}_i$ (i=1, 2, …, n) are the *normal modes* (or *natural modes, mode shapes,* etc.).

Properties of the Normal Modes:

Normal modes satisfy the following properties:

$$\overline{\mathbf{u}}_i^T \mathbf{K} \overline{\mathbf{u}}_j = 0, \qquad \overline{\mathbf{u}}_i^T \mathbf{M} \overline{\mathbf{u}}_j = 0, \qquad \text{for } i \neq j, \tag{7.16}$$

if $\omega_i \neq \omega_j$. That is, modes are *orthogonal* (thus *independent*) to each other with respect to **K** and **M** matrices.

Normal modes are usually normalized such that:

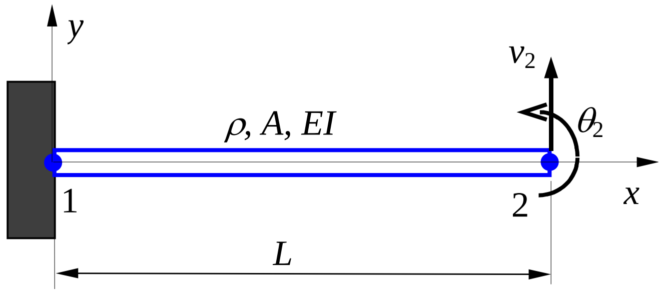
$$\overline{\mathbf{u}}_i^T \mathbf{M} \overline{\mathbf{u}}_i = 1, \qquad \overline{\mathbf{u}}_i^T \mathbf{K} \overline{\mathbf{u}}_i = \omega_i^2. \tag{7.17}$$

Notes:

- Magnitudes of displacements (modes) or stresses in normal mode analysis have no physical meanings.
- For normal mode analysis, no support of the structure is necessary.
- $\omega_i = 0$ means there are rigid-body motions of the whole or a part of the structure. This can be applied to check the FEA model (to see if there are rigid-body motions, mechanisms or free elements in the FEA models).
- Lower modes are more accurate than higher modes in the FEA calculations (because of less spatial variations in the lower modes, leading to that fewer elements/wave length are needed).

Example 7.1:

Consider the free vibration of a cantilever beam with one element as shown below.



We have the following equation for the free vibration (EVP):

$$\left[\mathbf{K}-\omega^2\mathbf{M}\right]\begin{Bmatrix}\overline{v}_2\\ \overline{\theta}_2\end{Bmatrix}=\begin{Bmatrix}0\\0\end{Bmatrix},$$

where

$$\mathbf{K}=\frac{EI}{L^3}\begin{bmatrix}12 & -6L\\ -6L & 4L^2\end{bmatrix}, \qquad \mathbf{M}=\frac{\rho AL}{420}\begin{bmatrix}156 & -22L\\ -22L & 4L^2\end{bmatrix}.$$

The equation for determine the natural frequencies is:

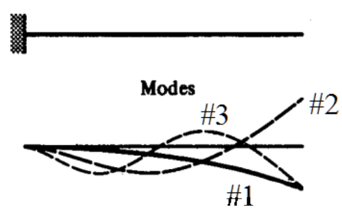
$$\begin{vmatrix}12-156\lambda & -6L+22L\lambda\\ -6L+22L\lambda & 4L^2-4L^2\lambda\end{vmatrix}=0,$$

in which $\lambda=\omega^2\rho AL^4/420EI$.

Solving the EVP, we obtain:

$$\omega_1=3.533\left(\frac{EI}{\rho AL^4}\right)^{1/2}, \qquad \begin{Bmatrix}\overline{v}_2\\ \overline{\theta}_2\end{Bmatrix}_1=\begin{Bmatrix}1\\ 1.38/L\end{Bmatrix},$$

$$\omega_2=34.81\left(\frac{EI}{\rho AL^4}\right)^{1/2}, \qquad \begin{Bmatrix}\overline{v}_2\\ \overline{\theta}_2\end{Bmatrix}_2=\begin{Bmatrix}1\\ 7.62/L\end{Bmatrix}.$$



The exact solutions of the first two natural frequencies for this problem are:

$$\omega_1=3.516\left(\frac{EI}{\rho AL^4}\right)^{1/2}, \qquad \omega_2=22.03\left(\frac{EI}{\rho AL^4}\right)^{1/2}.$$

We can see that for the FEA solution with one beam element, mode 1 is calculated much more accurately than mode 2. More elements are needed in order to compute mode 2 more accurately. The first three mode shapes of the cantilever beam is shown in the insert above.

III. Damping

There are two commonly used models for viscous damping.

A. Proportional Damping (Rayleigh Damping)

In this damping model, the damping matrix **C** is assumed to be proportional to the stiffness and mass matrices in the following fashion:

$$\mathbf{C} = \alpha \mathbf{K} + \beta \mathbf{M}, \tag{7.18}$$

where the constants α and β are found from the following two equations:

$$\xi_1 = \frac{\alpha\omega_1}{2} + \frac{\beta}{2\omega_1}, \qquad \xi_2 = \frac{\alpha\omega_2}{2} + \frac{\beta}{2\omega_2}, \tag{7.19}$$

with ω_1, ω_2, ξ_1 and ξ_2 (damping ratios) being specified by the user. The plots of the above two equations are shown in Figure 7.7.

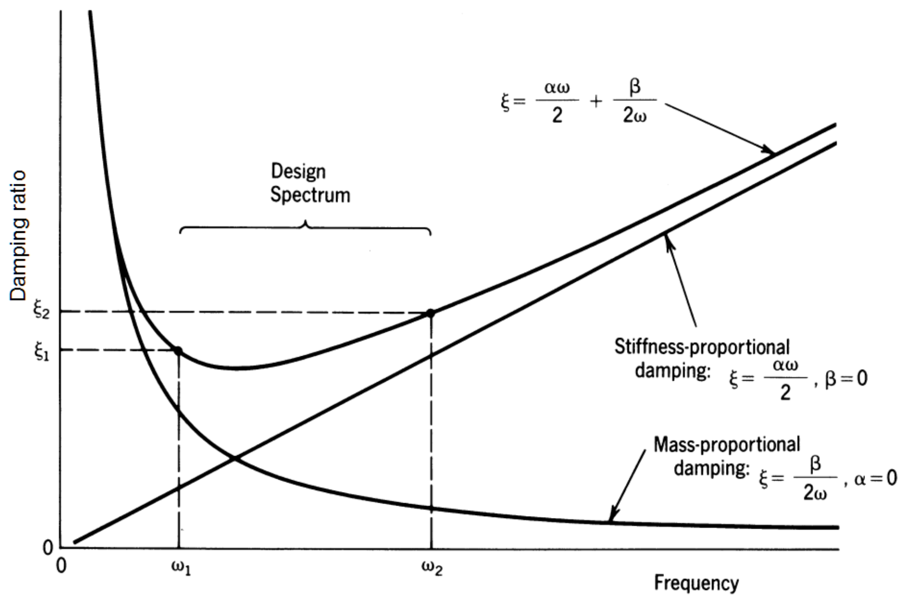


Figure 7.7. Two equations for determining the proportional damping coefficients.

B. Modal Damping

In this damping model, the viscous damping is incorporated in the modal equations to be discussed in the next section.

IV. Modal Equations

Use the normal modes (modal matrices), we can transform the coupled system of dynamic equations to uncoupled system of equations or modal equations.

We have:

$$\left[\mathbf{K} - \omega_i^2 \mathbf{M}\right] \overline{\mathbf{u}}_i = \mathbf{0}, \qquad i = 1, 2, \ldots, n, \tag{7.20}$$

where the normal modes $\overline{\mathbf{u}}_i$ satisfy:

$$\begin{cases} \overline{\mathbf{u}}_i^T \mathbf{K} \overline{\mathbf{u}}_j = 0, \\ \overline{\mathbf{u}}_i^T \mathbf{M} \overline{\mathbf{u}}_j = 0, \end{cases} \qquad \text{for } i \neq j,$$

and

$$\begin{cases} \overline{\mathbf{u}}_i^T \mathbf{M} \overline{\mathbf{u}}_i = 1, \\ \overline{\mathbf{u}}_i^T \mathbf{K} \overline{\mathbf{u}}_i = \omega_i^2, \end{cases} \qquad \text{for } i = 1, 2, \ldots, n.$$

Form the *modal matrix*:

$$\mathbf{\Phi}_{(n \times n)} = \left[\overline{\mathbf{u}}_1 \ \overline{\mathbf{u}}_2 \cdots \overline{\mathbf{u}}_n\right] \tag{7.21}$$

We can verify that:

$$\begin{aligned} &\mathbf{\Phi}^T \mathbf{K} \mathbf{\Phi} = \mathbf{\Omega} = \begin{bmatrix} \omega_1^2 & 0 & \cdots & 0 \\ 0 & \omega_2^2 & & \vdots \\ \vdots & & \ddots & 0 \\ 0 & \cdots & 0 & \omega_n^2 \end{bmatrix} \text{(Spectral matrix)}, \\ &\mathbf{\Phi}^T \mathbf{M} \mathbf{\Phi} = \mathbf{I}. \end{aligned} \tag{7.22}$$

Transformation for the displacement vector:

$$\mathbf{u} = z_1 \overline{\mathbf{u}}_1 + z_2 \overline{\mathbf{u}}_2 + \cdots + z_n \overline{\mathbf{u}}_n = \mathbf{\Phi} \mathbf{z}, \tag{7.23}$$

where

$$\mathbf{z} = \begin{Bmatrix} z_1(t) \\ z_2(t) \\ \vdots \\ z_n(t) \end{Bmatrix}$$

are called the *principal coordinates*.

Substitute (7.23) into the dynamic equation (7.8) and obtain:

$$\mathbf{M}\mathbf{\Phi}\ddot{\mathbf{z}} + \mathbf{C}\mathbf{\Phi}\dot{\mathbf{z}} + \mathbf{K}\mathbf{\Phi}\mathbf{z} = \mathbf{f}(t).$$

Pre-multiply this result by $\boldsymbol{\Phi}^T$, and apply (7.22):

$$\ddot{\mathbf{z}} + \mathbf{C}_{\phi}\dot{\mathbf{z}} + \mathbf{\Omega}\mathbf{z} = \mathbf{p}(t), \tag{7.24}$$

where $\mathbf{C}_{\phi} = \alpha\mathbf{I} + \beta\mathbf{\Omega}$ if proportional damping is applied, and $\mathbf{p} = \mathbf{\Phi}^T\mathbf{f}(t)$.

If we introduce *modal damping*:

$$\mathbf{C}_{\phi} = \begin{bmatrix} 2\xi_1\omega_1 & 0 & \cdots & 0 \\ 0 & 2\xi_2\omega_2 & & \\ \vdots & & \ddots & \vdots \\ 0 & & \cdots & 2\xi_n\omega_n \end{bmatrix}, \tag{7.25}$$

where ξ_i is the damping ratio at mode *i*, Eq. (7.24) becomes:

$$\ddot{z}_i + 2\xi_i\omega_i\dot{z}_i + \omega_i^2 z_i = p_i(t), \qquad i = 1, 2, \ldots, n. \tag{7.26}$$

Equations in (7.24) with modal damping, or in (7.26), are called *modal equations*. These equations are uncoupled, second-order differential equations, which are much easier to solve than the original dynamic equation which is a coupled system.

To recover **u** from **z,** apply transformation (7.23) again, once **z** is obtained from (7.26).

Notes:

- Only the first few modes may be needed in constructing the modal matrix $\mathbf{\Phi}$ (that is, $\mathbf{\Phi}$ could be an $n\times m$ rectangular matrix with $m<n$). Thus, significant reduction in the size of the system can be achieved.
- Modal equations are best suited for structural vibration problems in which higher modes are not important (that is, for structural vibrations, but not for structures under impact or shock loadings).

V. Frequency Response Analysis

Frequency response analysis is also called harmonic response analysis, when the applied dynamic load is a sine or cosine functions. In this case, the equation of motion is:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \underbrace{\mathbf{F}\sin\omega t}_{\text{Harmonic loading}} \tag{7.27}$$

A. Modal Method

In this approach, we apply the modal equations, that is:

$$\ddot{z}_i + 2\xi_i\omega_i\dot{z}_i + \omega_i^2 z_i = p_i \sin\omega t, \quad i = 1, 2, \ldots, m. \tag{7.28}$$

These are uncoupled equations. The solutions for **z** are in the form:

$$z_i(t) = \frac{p_i / \omega_i^2}{\sqrt{(1-\eta_i^2)^2 + (2\xi_i \eta_i)^2}} \sin(\omega t - \theta_i), \tag{7.29}$$

where

$$\begin{cases} \theta_i = \arctan \dfrac{2\xi_i \eta_i}{1-\eta_i^2}, & \text{phase angle;} \\ \eta_i = \omega / \omega_i; & \\ \xi_i = \dfrac{c_i}{c_c} = \dfrac{c_i}{2m\omega_i}, & \text{damping ratio.} \end{cases}$$

The response of each mode z_i is similar to that of a single DOF system. Once the natural coordinate vector $\mathbf{z}$ is known, we can recover the real displacement vector $\mathbf{u}$ from $\mathbf{z}$ using Eq. (7.23).

B. Direct Method

In this approach, we solve Eq. (7.27) directly, that is, compute the inverse of the coefficient matrix, which is in general much more expensive than the modal method.

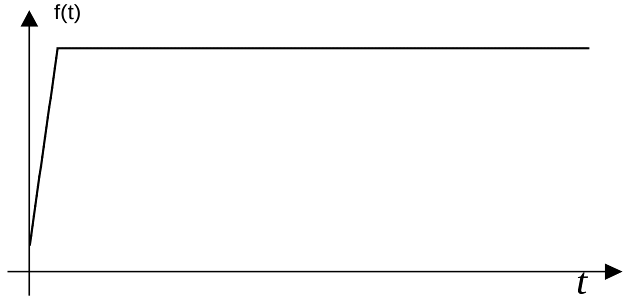
Using complex notation to represent the harmonic response, we have $\mathbf{u} = \bar{\mathbf{u}} e^{i\omega t}$ and Eq. (7.27) becomes:

$$\left[\mathbf{K} + i\omega\mathbf{C} - \omega^2\mathbf{M}\right]\bar{\mathbf{u}} = \bar{\mathbf{F}} \tag{7.30}$$

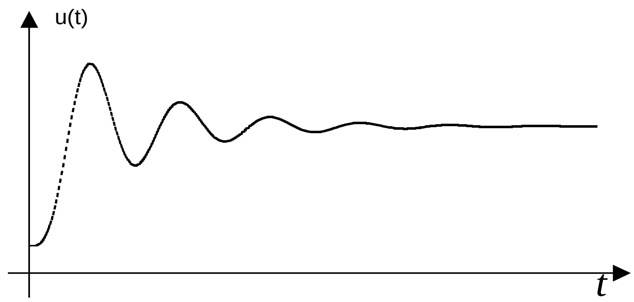
Inverting the matrix $\left[\mathbf{K} + i\omega\mathbf{C} - \omega^2\mathbf{M}\right]$, we can obtain the displacement amplitude vector $\bar{\mathbf{u}}$. However, this equation is expensive to solve for large systems and the matrix $\left[\mathbf{K} + i\omega\mathbf{C} - \omega^2\mathbf{M}\right]$ can become ill-conditioned if ω is close to any natural frequency ω_{i} of the structure. Therefore, the direct method is only applied when the system of equations is small and the frequency is away from any natural frequency of the structure.

VI. Transient Response Analysis

In the transient response analysis, also called dynamic response/time-history analysis, we are interested in computing the responses of the structures under *arbitrary* time-dependent loading (Figure 7.8).



(a)



(b)

Figure 7.8. (a) A step type of loading; (b) Structural response to the step loading.

To compute the transient responses, integration through time is employed (Figure 7.9).

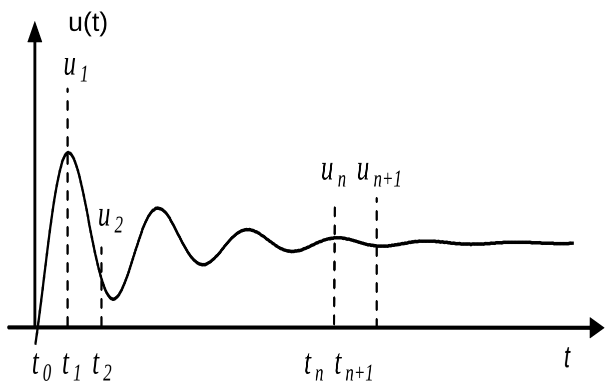


Figure 7.9. Computing the responses by integration through time.

We write the equation of motion at instance t_n, $n = 0, 1, 2, 3, \cdots$, as:

$$\mathbf{M}\ddot{\mathbf{u}}_n + \mathbf{C}\dot{\mathbf{u}}_n + \mathbf{K}\mathbf{u}_n = \mathbf{f}_{n.} \tag{7.31}$$

Then, we introduce time increments: $\Delta t = t_{n+1} - t_n$, n=0, 1, 2, 3, ···, and integrate through the time.

There are two categories of methods for transient analysis as described in the following sections.

A. Direct Methods (Direct Integration Methods)

Central Difference Method:

Approximate the velocity and acceleration vectors by using finite difference:

$$\begin{aligned}\dot{\mathbf{u}}_n &= \frac{1}{2\Delta t}(\mathbf{u}_{n+1} - \mathbf{u}_{n-1}),\\ \ddot{\mathbf{u}}_n &= \frac{1}{(\Delta t)^2}(\mathbf{u}_{n+1} - 2\mathbf{u}_n + \mathbf{u}_{n-1})\end{aligned} \tag{7.32}$$

Dynamic equation becomes,

$$\mathbf{M}\left[\frac{1}{(\Delta t)^2}(\mathbf{u}_{n+1} - 2\mathbf{u}_n + \mathbf{u}_{n-1})\right] + \mathbf{C}\left[\frac{1}{2\Delta t}(\mathbf{u}_{n+1} - \mathbf{u}_{n-1})\right] + \mathbf{K}\mathbf{u}_n = \mathbf{f}_n,$$

which yields

$$\mathbf{A}\mathbf{u}_{n+1} = \mathbf{F}(t) \tag{7.33}$$

where

$$\begin{cases}\mathbf{A} = \dfrac{1}{(\Delta t)^2}\mathbf{M} + \dfrac{1}{2\Delta t}\mathbf{C},\\[2ex] \mathbf{F}(t) = \mathbf{f}_n - \left[\mathbf{K} - \dfrac{2}{(\Delta t)^2}\mathbf{M}\right]\mathbf{u}_n - \left[\dfrac{1}{(\Delta t)^2}\mathbf{M} - \dfrac{1}{2\Delta t}\mathbf{C}\right]\mathbf{u}_{n-1}.\end{cases}$$

We compute u_{n+1} from u_n and u_{n-1}, which are known from the previous time step. The solution procedure is repeated or marching from $t_0, t_1, \cdots t_n, t_{n+1}, \cdots,$ until reach the specified maximum time. This method is unstable if Δt is too large.

Newmark Method:

We use the following approximations:

$$\begin{aligned}\mathbf{u}_{n+1} &\approx \mathbf{u}_n + \Delta t\dot{\mathbf{u}}_n + \frac{(\Delta t)^2}{2}\left[(1-2\beta)\ddot{\mathbf{u}}_n + 2\beta\ddot{\mathbf{u}}_{n+1}\right], \rightarrow (\ddot{\mathbf{u}}_{n+1} = \cdots)\\ \dot{\mathbf{u}}_{n+1} &\approx \dot{\mathbf{u}}_n + \Delta t\left[(1-\gamma)\ddot{\mathbf{u}}_n + \gamma\ddot{\mathbf{u}}_{n+1}\right],\end{aligned} \tag{7.34}$$

where β and γ are chosen constants. These lead to the following equation:

$$\mathbf{A}\mathbf{u}_{n+1} = \mathbf{F}(t) \tag{7.35}$$

where

$$\mathbf{A} = \mathbf{K} + \frac{\gamma}{\beta \Delta t}\mathbf{C} + \frac{1}{\beta (\Delta t)^2}\mathbf{M},$$
$$\mathbf{F}(t) = f(\mathbf{f}_{n+1}, \gamma, \beta, \Delta t, \mathbf{C}, \mathbf{M}, \mathbf{u}_n, \dot{\mathbf{u}}_n, \ddot{\mathbf{u}}_n).$$

This method is unconditionally stable if

$$2\beta \geq \gamma \geq \frac{1}{2}.$$

For example, we can use $\gamma = \frac{1}{2}$, $\beta = \frac{1}{4}$, which gives the constant average acceleration method.

Direct methods can be expensive, because of the need to compute A^{-1}, repeatedly for each time step if nonuniform time steps are used.

B. Modal Method

In this method, we first do the transformation of the dynamic equations using the modal matrix before the time marching:

$$\begin{aligned} &\mathbf{u} = \sum_{i=1}^{m} \overline{\mathbf{u}}_i z_i(t) = \Phi \mathbf{z}, \\ &\ddot{z}_i + 2\xi_i \omega_i \dot{z}_i + \omega_i z_i = p_i(t), \end{aligned} \qquad i = 1,2,\cdots, m. \tag{7.36}$$

Then, solve the uncoupled equations using an integration method. We can use, for example, 10% of the total modes ($m = n/10$). The advantages of the modal method are as follows:

- Uncoupled system;
- Fewer equations;
- No inverse of matrices;
- More efficient for large problems.

However, the modal method is less accurate if higher modes are important, which is the case for structures under impact or shock loading. Table 7.2 summarizes the advantages and disadvantages of the direct and modal methods for transient response analysis.

Table 7.2. Comparisons of the methods

Direct Methods	*Modal Method*
• Small models	• Large models
• More accurate (with small Δt)	• Higher modes ignored
• Single loading	• Multiple loading
• Shock loading	• Periodic loading
• …	• …

Cautions in Dynamic Analysis

- Symmetry model should not be used in the dynamic analysis (normal modes, etc.) because symmetric structures can have non-symmetric modes. However, symmetry can still be applied in creating the FEA model of a symmetric structure.
- Mechanism or rigid body motion means $\omega = 0$. Can use this to check FEA models to see if they are properly connected and/or supported.
- Input for FEA: Loading $F(t)$ or $F(\omega)$ can be very complex and data can be enormous in real engineering applications (for example, the load data for a car) and thus they often need to be filtered first before being used as input for FEA.

Example 7.2:

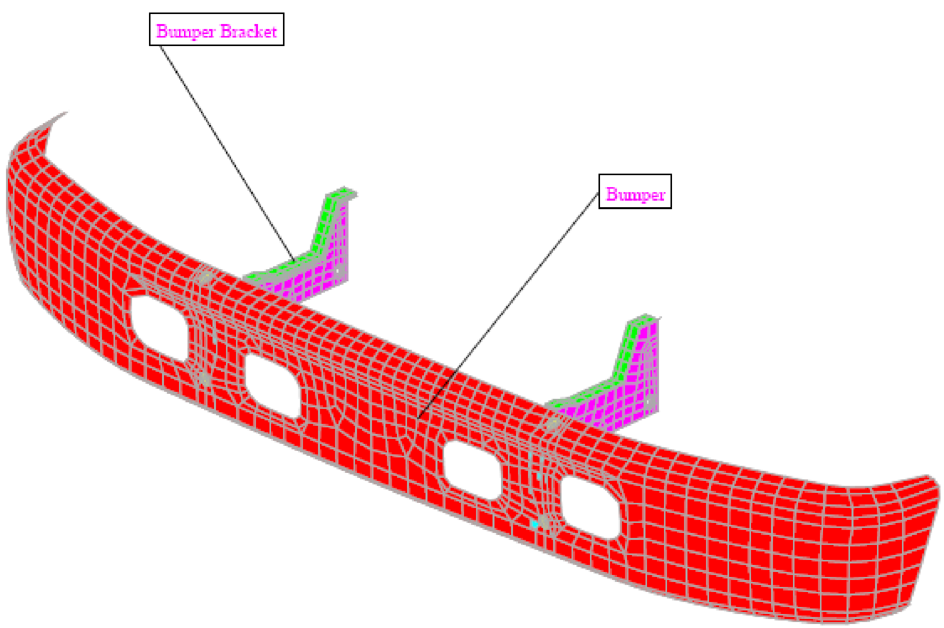


Figure 7.10. FEA model of a front bumper and supporting brackets.

Figure 7.10 shows a front bumper and the supporting brackets in a car. The model was applied to study the dynamic responses of the bumper. Shell elements were used for this study and the natural frequencies and vibration modes were obtained first. Figure 7.11 shows the first mode of the bumper when it is constrained at the bracket locations. The frequency response of the bumper was also analyzed using the same FE model, as shown by the red curve in Figure 7.12 and with the acceleration of the two brackets as the input. Several modifications of the bumper design were also studied with the goal to increase the base natural frequency (for example, from below 30 Hz to above 35 Hz) and to reduce the magnitudes of the frequency responses. The improved responses are shown by the other three curves in Figure 7.12.

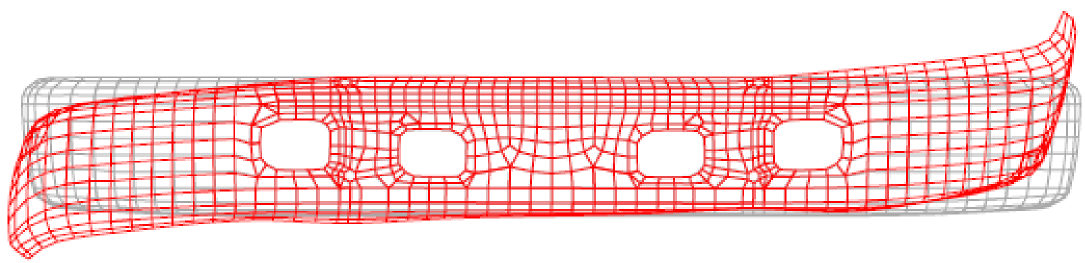


Figure 7.11. The first vibration mode of the bumper.

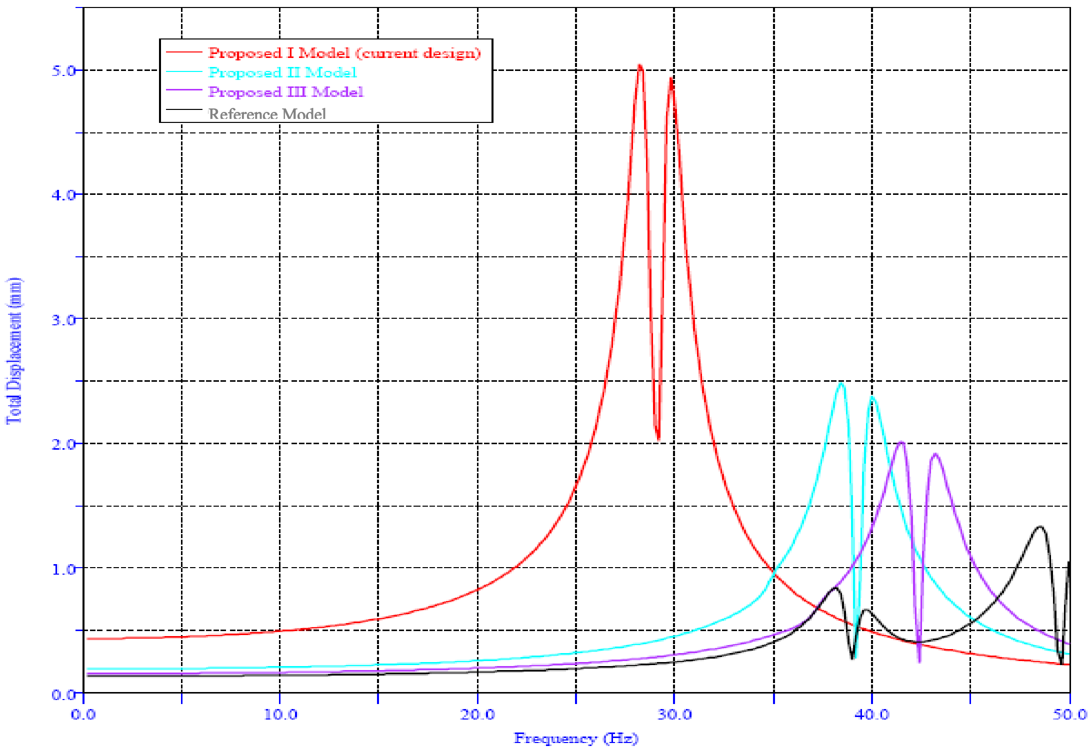


Figure 7.12. Frequency response of the bumper from 0 to 50 Hz.

Example 7.3:

One of the most interesting applications of the dynamic analysis with FEA is to conduct crash analysis and virtual drop tests of various products. The most popular FEA software for such analyses is the LS-DYNA package from Livermore Software Technology Corporation (LSTC). Figure 7.13 is an example of crash analysis of a car using LS-DYNA. Figure 1.4 is an example of drop test simulation of a soda can, also using LS-DYNA.

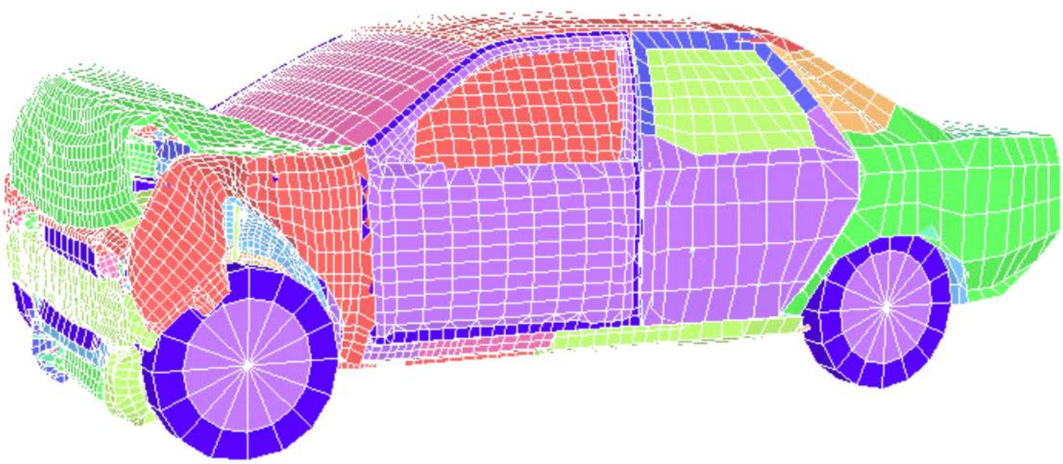


Figure 7.13. Car crash analysis using LS-DYNA (from LSTC website www.lstc.com).

More information about how to perform the impact analysis and drop test using the dynamic FEA, especially the software LS-DYNA, can be found from the LSTC website and the documents with the LS-DYNA software package.

VII. Summary

In this chapter, we first reviewed the equation of motion for both single DOF and multiple DOF systems and discussed how to compute the mass and damping matrices in the FEA formulations. Then, we discussed the methods for solving normal modes, harmonic responses, and transient responses for structural vibration and dynamic problems. The advantages and disadvantages of the direct method and modal method are discussed. Several examples of vibration and dynamic analyses are also discussed to show the applications of the FEA in vibration and dynamic analyses.

VIII. Problems

Problem 1. For the cantilever beam studied in Example 7.1, apply more beam elements and investigate the convergence of the FEA solutions for the first ten natural frequencies and normal modes of the beam.

Problem 2. For a square plate with edge length = 1 m, thickness = 0.005 m, Young's modulus E = 70 GPa, Poisson's ratio $\upsilon = 0.3$, and mass density $\rho = 2800$ kg/m^3, find the first five natural frequencies and normal modes using the FEA when the plate is:

(a) Clamped at the four edges;

(b) Simply supported at the four edges;

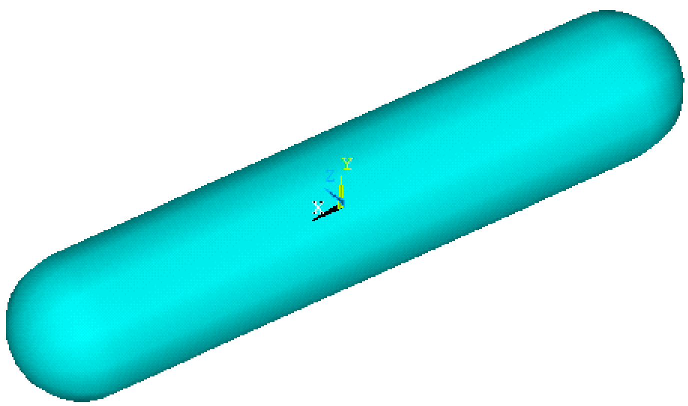
(c) Free at the four edges (not supported).

Study the convergence of your FEA results and discuss the effects of the support on the natural frequencies of the plate.

Problem 3. A fuel tank, with a total length = 5 m, diameter = 1 m, and thickness = 0.01 m, is shown below. Using the FEA, find the first ten natural frequencies and corresponding normal modes, when:

(a) The tank is not supported at all;

(b) The tank is constrained along the circumferences in the radial directions at the two locations 1 m away from the two ends.



Carefully build your FE mesh (using shell elements) so that the symmetry of the tank is reserved and the boundary conditions in part (b) can be applied readily. Assume the tank is made of steel with the Young's modulus E = 200 GPa, Poisson's ratio $\upsilon = 0.3$, and mass density $\rho = 7850$ kg/m^3.

Chapter 8. Thermal Analysis

In this chapter, we will discuss briefly the thermal analysis using the FEA. Thermal stresses due to changes of the temperatures are common in most engineering systems, such as cars, airplanes, bridges, electronic devices, and many consumer products.

The two main objectives in thermal analysis are:

- Determine the temperature field (steady state or unsteady state);
- Determine the thermal stresses in structures due to the temperature changes.

I. Temperature Field

For the temperature field in a 1-D space, such as a bar (Figure 8.1), we have the following Fourier heat conduction equation:

$$f_x = -k\frac{\partial T}{\partial x}, \tag{8.1}$$

where,

f_x = heat flux per unit area,

k = thermal conductivity,

$T = T(x, t)$ = temperature field.

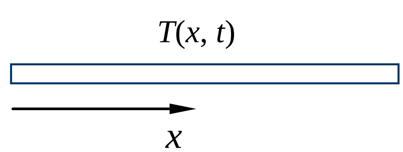


Figure 8.1. The temperature field $T(x, t)$ in a 1-D bar model.

For 3-D case, we have:

$$\begin{Bmatrix} f_x \\ f_y \\ f_z \end{Bmatrix} = -\mathbf{K}\begin{Bmatrix} \partial T/\partial x \\ \partial T/\partial y \\ \partial T/\partial z \end{Bmatrix}, \tag{8.2}$$

where, f_x, f_y, f_z = heat flux in the x, y and z direction, respectively. In the case of isotropic materials, the conductivity matrix is:

$$\mathbf{K} = \begin{bmatrix} k & 0 & 0 \\ 0 & k & 0 \\ 0 & 0 & k \end{bmatrix}. \tag{8.3}$$

The equation of heat flow is given by:

$$-\left[\frac{\partial f_x}{\partial x}+\frac{\partial f_y}{\partial y}+\frac{\partial f_z}{\partial z}\right]+q_v=c\rho\frac{\partial T}{\partial t} \tag{8.4}$$

in which,

q_v = rate of internal heat generation per unit volume,

c = specific heat,

ρ = mass density.

For steady state case ($\partial T/\partial t=0$) and isotropic materials, we can obtain:

$$k\nabla^2 T=-q_v\,. \tag{8.5}$$

This is a Poisson equation, which needs to be solved under given boundary conditions.

Boundary conditions for steady state heat conduction problems are (Figure 8.2):

$$T=\overline{T}, \qquad \text{on } S_T\,; \tag{8.6}$$

$$Q\equiv -k\frac{\partial T}{\partial n}=\overline{Q}, \qquad \text{on } S_q\,. \tag{8.7}$$

Note that at any point on the boundary $S=S_T\cup S_q$, only one type of BCs can be specified.

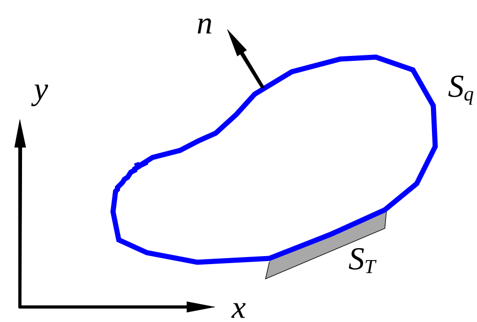


Figure 8.2. Boundary conditions for heat conduction problems.

Finite Element Formulation for Heat Conduction:

For heat conduction problems, we can establish the following FE equation:

$$\mathbf{K}_T\mathbf{T}=\mathbf{q} \tag{8.8}$$

where,

$\mathbf{K}_T$ = conductivity matrix,

$\mathbf{T}$ = vector of nodal temperature,

$\mathbf{q}$ = vector of thermal loads.

The element conductivity matrix is given by:

$$\mathbf{k}_T = \int_V \mathbf{B}^T \mathbf{K} \mathbf{B} dV . \tag{8.9}$$

This is obtained in a similar way as for the structural analysis, that is, by starting with the interpolation $T = \mathbf{N}\mathbf{T}_e$ for the temperature field (with $\mathbf{N}$ being the shape function matrix and T_e the nodal temperature). Note that there is only one DOF at each node for the thermal problems.

For transient (unsteady state) heat conduction problems, we have:

$$\frac{\partial T}{\partial t} \neq 0 .$$

In this case, we need to apply finite difference schemes (use time steps and integrate in time), as in the transient structural analysis, to obtain the transient temperature fields.

II. Thermal Stress Analysis

To determine the thermal stresses due to temperature changes in structures, we can proceed to:

- Solve Eq. (8.8) first to obtain the temperature (change) fields.
- Apply the temperature change ΔT as initial strains (or initial stresses) to the structure to compute the thermal stresses due to the temperature change.

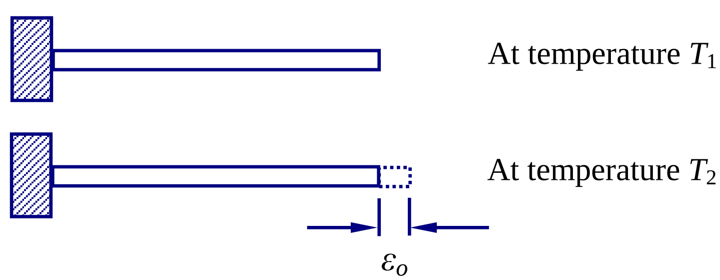


Figure 8.3. Expansion of a bar due to increase in temperature.

1-D Case:

To understand the stress-strain relations in cases of solids undergo temperature changes, we first examine the 1-D case (Figure 8.3). We have for the thermal strain (or initial strain):

$$\varepsilon_o = \alpha \Delta T , \tag{8.10}$$

in which,

α = the coefficient of thermal expansion,

$\Delta T = T_2 - T_1$ = change of temperature.

Total strain is given by:

$$\varepsilon = \varepsilon_e + \varepsilon_o \tag{8.11}$$

with ε_e being the elastic strain due to mechanical load.

That is, the total strain can be written as:

$$\varepsilon = E^{-1}\sigma + \alpha\Delta T\,, \tag{8.12}$$

Or, inversely, the stress is given by:

$$\sigma = E(\varepsilon - \varepsilon_o)\,. \tag{8.13}$$

Example 8.1:

Consider the bar under thermal load ΔT as shown in Figure 8.3.

(a) If no constraint on the right-hand side, that is, the bar is free to expand to the right, then we have:

$$\varepsilon = \varepsilon_o,\ \ \varepsilon_e = 0,\ \ \sigma = 0\,,$$

from Eq. (8.13), that is, there is no thermal stress in this case!

(b) If there is a constraint on the right-hand side, that is, the bar can not expand to the right, then we have:

$$\varepsilon = 0,\quad \varepsilon_e = -\varepsilon_o = -\alpha\Delta T, \qquad \sigma = -E\alpha\Delta T\,,$$

from Eqs. (8.11) and (8.13). Thus, thermal stress exists!

From this simple example, we see that the way in which the structure is constrained has a critical role in inducing the thermal stresses.

2-D Cases:

For plane stress, we have:

$$\boldsymbol{\varepsilon}_o = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}_o = \begin{Bmatrix} \alpha\Delta T \\ \alpha\Delta T \\ 0 \end{Bmatrix}. \tag{8.14}$$

For plane strain, we have:

$$\varepsilon_o = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}_o = \begin{Bmatrix} (1+\nu)\alpha\Delta T \\ (1+\nu)\alpha\Delta T \\ 0 \end{Bmatrix}, \tag{8.15}$$

in which, ν is the Poisson's ratio.

3-D Case:

$$\boldsymbol{\varepsilon}_o = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}_o = \begin{Bmatrix} \alpha\Delta T \\ \alpha\Delta T \\ \alpha\Delta T \\ 0 \\ 0 \\ 0 \end{Bmatrix}. \tag{8.16}$$

Observation: Temperature changes do not yield shear strains.

In both 2-D and 3-D cases, the total strain can be given by the following vector equation:

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}_e + \boldsymbol{\varepsilon}_o. \tag{8.17}$$

And the stress-strain relation is given by:

$$\boldsymbol{\sigma} = \mathbf{E}\boldsymbol{\varepsilon}_e = \mathbf{E}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_o). \tag{8.18}$$

Notes on FEA for Thermal Stress Analysis:

- Need to specify α for the structure and ΔT on the related elements (which experience the temperature change).
- Note that for linear thermoelasticity, same temperature change will yield same stresses, even if the structure is at two different temperature levels.
- Differences in the temperatures during the manufacturing and working environment are the main cause of thermal (residual) stresses.

Example 8.2:

First, we study a heat sink model taken from Ref. [8] for thermal analysis. A heat sink is a device commonly used to dissipate heat from a CPU in a computer. In this heat sink model, a given temperature field ($T = 120$) is specified on the bottom surface and a heat flux condition ($Q \equiv -k\dfrac{\partial T}{\partial n} = -0.2$) is specified on all the other surfaces. The 20-node brick elements are used and the FE mesh shown in Figure 8.4 has 127,149 nodes. This mesh for the volume was obtained by extruding the cross section meshed with quadrilateral area elements so that a mapped mesh was obtained. The computed temperature distribution on the heat sink using ANSYS is shown in Figure 8.5. The cooling effect of the heat sink is most evident.

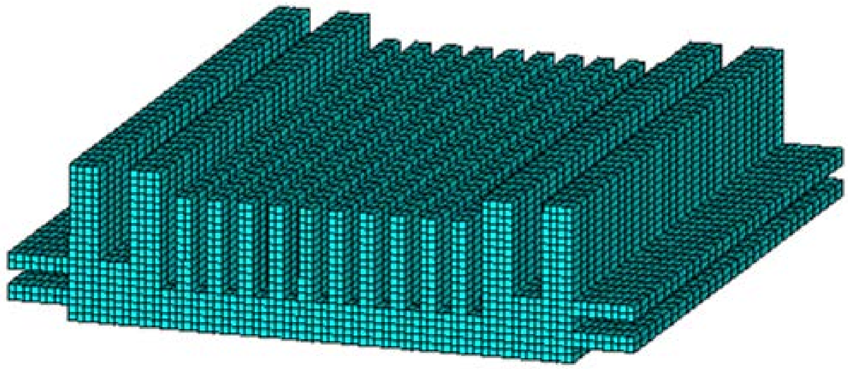


Figure 8.4. A heat-sink model used for heat conduction analysis.

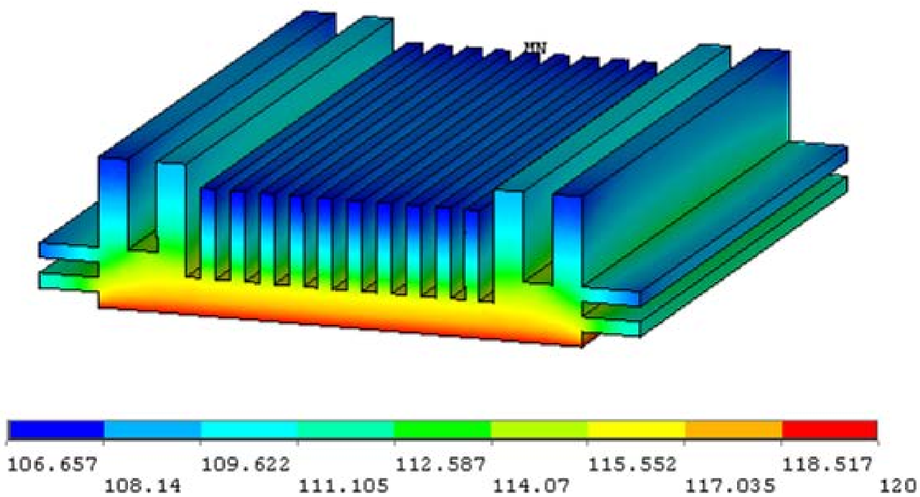


Figure 8.5.Computed temperature distribution in the heat-sink.

Example 8.3:

Next, we study the thermal stresses in structures due to temperature changes. For this purpose, we employ the same model of a plate with a center hole (Figure 8.6) as used in Chapter 3 and Chapter 4 to show the relation between the thermal stresses and constraints. We assume that the plate is made of steel with the Young's modulus $E = 200$ GPa, Poisson's ratio $\nu = 0.3$ and thermal expansion coefficient $\alpha = 12\text{x}10^{-6}$ 1/°C. The plate is applied with a uniform temperature increase of 100 °C.

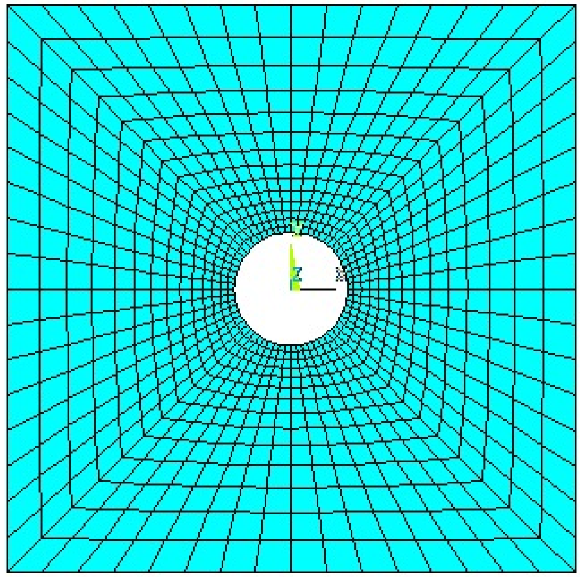
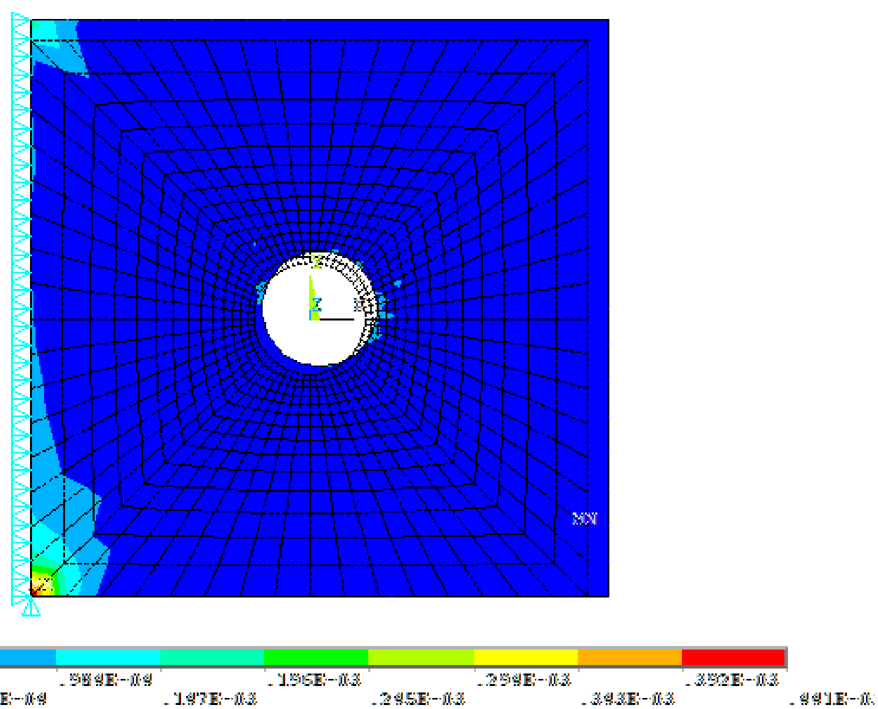


Figure 8.6. A square plate with a center hole and under a uniform temperature load.

Figure 8.7 shows the computed thermal stresses in the plate under two different types of constraints. When the plate is constrained (roller support) at the left side only, the plate expands uniformly in both the *x* and *y* directions, which causes no thermal stresses (Figure 8.7 (a), note that the numbers, ranging from 10^{-6} to 10^{-3}, are actually machine zeros). However, when the plate is constrained at both the left and right sides, the plate can expand only in the *y* direction and significant thermal stresses are induced (Figure 8.7 (b)), especially near the edge of the hole.

(a)



(b)

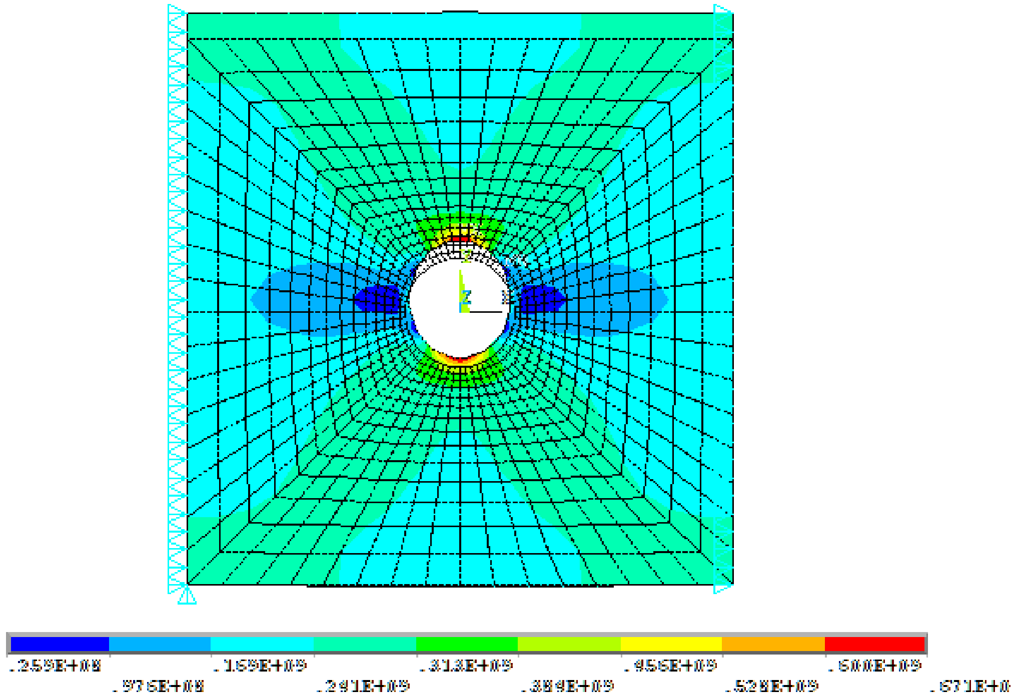


Figure 8.7. Thermal (von Mises) stresses in the plate: (a) When the plate is constrained at left side only (thermal stresses = 0); (b) When the plate is constrained at both left and right sides.

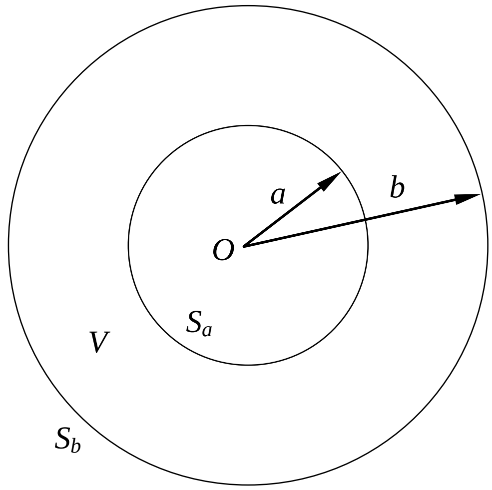
In many cases, the changes of the material properties of a structure should be considered as well when the temperature changes are significant, especially when the structure is exposed to high temperatures such as in an aircraft engine. Cyclic temperature fields can also cause thermal fatigue of structures and lead to failures. All these phenomena can be modeled with the FEA and interested readers can consult with the documents of the FEA software at hand.

III. Summary

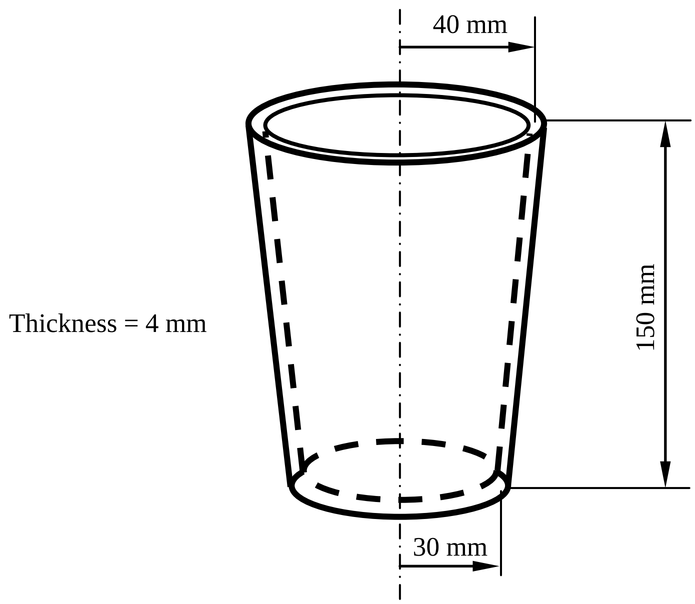
In this chapter, we briefly discussed the governing equations for heat conduction problems and the FEA formulation. Thermal stresses due to changes of temperatures in structures are also discussed and the effects of constraints of the structures on the thermal stresses are emphasized.

IV. Problems

Problem 1. Study the heat conduction problem in a simple annular region shown below, using the FEA. Assume $a = 1$, $b = 2$, $T_a = 100$, and $Q_b = -k\dfrac{\partial T}{\partial n} = -200$. Determine the temperature field and heat flux in this region and compare your FEA results with the analytical solution.



Problem 2. For the same glass cup model studied in Chapter 5, as shown in the figure below, determine the thermal stresses when the inner surfaces of the cup experience a temperature change from a room temperature of 20 °C to 100 °C, while all other surfaces are kept at the same room temperature of 20 °C. Assume that the cup has a uniform thickness, E = 70 GPa, ν = 0.17 and the coefficient of thermal expansion $\alpha = 8.0\times10^{-6}$ / °C.



References for Further Studies

1. O. C. Zienkiewicz, "The birth of the finite element method and of computational mechanics," *International Journal for Numerical Methods in Engineering*, **60**, 3-10 (2004).
2. O. C. Zienkiewicz and R. L. Taylor, *The Finite Element Method*, 5th ed (McGraw-Hill, New York, 2000).
3. J. N. Reddy, *An Introduction to the Finite Element Method*, 3rd ed (McGraw-Hill, New York, 2006).
4. R. D. Cook, *Finite Element Modeling for Stress Analysis* (John Wiley & Sons, Inc., New York, 1995).
5. R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, *Concepts and Applications of Finite Element Analysis*, 4th ed (John Wiley & Sons, Inc., New York, 2002).
6. K. J. Bathe, *Finite Element Procedures* (Prentice Hall, Englewood Cliffs, NJ, 1996).
7. T. R. Chandrupatla and A. D. Belegundu, *Introduction to Finite Elements in Engineering*, 3rd ed (Prentice Hall, Upper Saddle River, NJ, 2002).
8. S. Moaveni, *Finite Element Analysis - Theory and Application with ANSYS*, 3rd ed (Prentice-Hall, Upper Saddle River, NJ, 2007).
9. Y. C. Fung, *A First Course in Continuum Mechanics*, 3rd ed (Prentice Hall, Englewood Cliffs, 1994).
10. P. Timoshenko and J. N. Goodier, *Theory of Elasticity*, 3rd ed (McGraw-Hill, New York, 1987).
11. S. P. Timoshenko and S. Woinowsky-Krieger, *Theory of Plates and Shells*, 2nd ed (McGraw-Hill, New York, 1987).