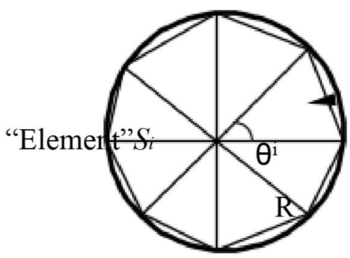
INTRODUCTION

BasicConcepts

The *finite element method* (FEM), or *finite element analysis*(FEA), is based on the idea of building a complicated object withsimple blocks, or, dividing a complicated object into small andmanageable pieces.Application of this simple idea can be foundeverywherein everyday life, as wellas inengineering.

Examples:

- Lego (kids' play)
- Buildings
- Approximation of the area of a circle:



Why Finite Element Method?

- Design analysis: hand calculations, experiments, andcomputer simulations
- FEM/FEA is the most widely applied computer simulation method in engineering
- Closely integrated with CAD/CAM applications

A Brief History of the FEM

- 1943 ----- Courant (Variational methods)
- 1956 ----- Turner,Clough,MartinandTopp(Stiffness)
- 1960 ----- Clough("FiniteElement",planeproblems)
- 1970s Applicationsonmainframecomputers
- 1980s Microcomputers,pre-andpostprocessors

- 1990s Analysisoflargestructuralsystems

What is FEM ?

- Many physical phenomena in engineering and science can be described in terms of *partial differential equations*.
- In general, solving these equations by classical analytical methods for arbitrary shapes is almost impossible.
- The *finite element method* (FEM) is a numerical approach by which these partial differential equations can be solved approximately.
- From an engineering standpoint, the FEM is a method for solving engineering problems such as *stress analysis, heat transfer, fluid flow* and *electromagnetics* by computer simulation.
- Millions of engineers and scientists worldwide use the FEM to predict the behaviour of structural, mechanical, thermal, electrical and chemical systems for both design and performance analyses.

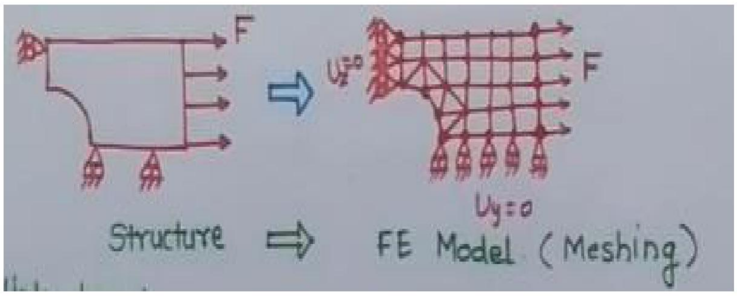
BASIC STEPS OF FEM

1 Discretization of the structure

2 Identify primary unknown quantity 3 Selection of Displacement function
4 Formation of the element stiffness matrix and load vector
5 Formation of Global stiffness matrix and load vector
6 Incorporation of Boundary conditions 7 Solution of Simultaneous equations
8 Calculation of element strains and stresses
9 Interpretation of the result obtained.

Step1: Discretization of or structure – (Establish the FE mesh)

- The continuum is divided into a number of elements by imaginary lines or surfaces.
- The interconnected elements may have different sizes and shapes.
- Establish the FE mesh with set coordinates, element numbers and node numbers
- The discretized FE model must be situated with a coordinate system
- Elements and nodes in the discretized FE model need to be identified by “element numbers” and “nodal numbers.”
- Nodes are identified by the assigned node numbers and their corresponding coordinates

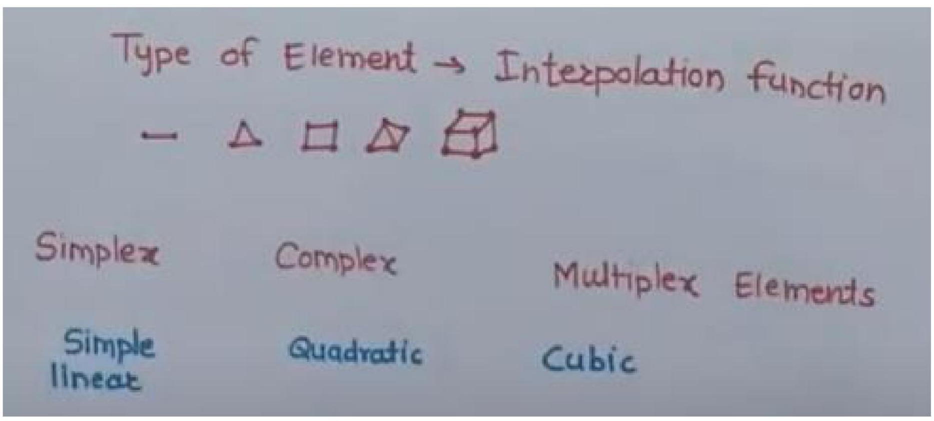


Step2: Identify primary unknown quantity

- Primary unknown quantity - The first and principal unknown quantity to be obtained by the FEM
  Eg: Stress analysis: Displacement {u} at nodes
- In stress analysis, The primary unknowns are nodal displacements, but secondary unknown quantities include: strains in elements can be obtained by the "strain-displacement relations," and the unknown stresses in the elements by the stress-strain relations (the Hooke's law).

Step 3: Choice of approximating functions

- Displacement function is the starting point of the mathematical analysis.
- This represents the variation of the displacement within the element.
- The displacement function may be approximated in the form a linear function or a higher-order function.
- A convenient way to express it is by polynomial expressions.
- Shape or geometry of the element may also be approximated.



Step 4: Formation of the element stiffness matrix & load vector

- After continuum is discretized with desired element shapes, the individual element stiffness matrix is formulated.
- Basically it is a minimization procedure whatever may be the approach adopted.
- For certain elements, the form involves a great deal of sophistication.
- The geometry of the element is defined in reference to the global frame.
- Coordinate transformation must be done for elements where it is necessary.

$$\{F\}e= \{K\}e * \{q\}e$$

Step 5: Formation of overall stiffness matrix & load vector

- After the element stiffness matrices in global coordinates are formed, they are assembled to form the overall stiffness matrix.
- The assembly is done through the nodes which are common to adjacent elements.
- The overall stiffness matrix is symmetric and banded.

$$\{F\}G= \{K\} G * \{q\} G$$

Step6 : Incorporation of boundary conditions

- The boundary restraint conditions are to be imposed in the stiffness matrix.
- There are various techniques available to satisfy the boundary conditions.
- One is the size of the stiffness matrix may be reduced or condensed in its final

form.

- To ease computer programming aspect and to elegantly incorporate the boundary conditions, the size of overall matrix is kept the same.

Step7: Solution of simultaneous equations

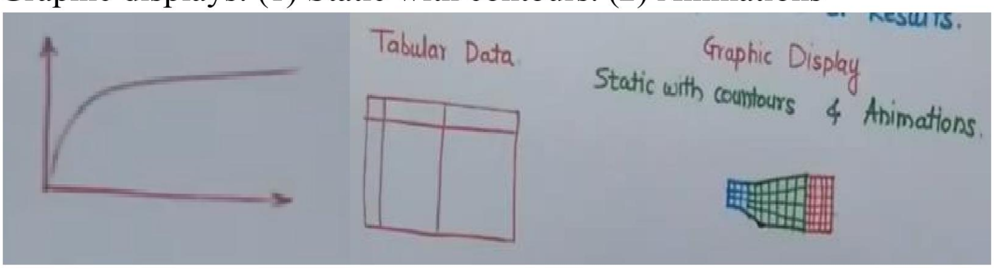
- The unknown nodal displacements are calculated by the multiplication of force vector with the inverse of stiffness matrix.
- [δ]=inverse of [k].[F]

Step8: Calculation of stresses or stress-resultants

- Nodal displacements are utilized for the calculation of stresses or stress-resultants.
- This may be done for all elements of the continuum or it may be limited to some predetermined elements.

Step9: Display and Interpretation of Results

- Results may also be obtained by graphical means.
- It may desirable to plot the contours of the deformed shape of the continuum.
- Tabulation of results
- Graphic displays: (1) Static with contours. (2) Animations



Advantages of Finite Element Method

- Modeling of complex geometries and irregular shapes are easier as varieties of finite elements are available for discretization of domain.
- Boundary conditions can be easily incorporated in FEM.
- Different types of material properties can be easily accommodated in modeling from element to element or even within an element.
- Higher order elements may be implemented.
- FEM is simple, compact and result-oriented and hence widely popular among engineering community.
- Availability of large number of computer software packages and literature makes FEM a versatile and powerful numerical method.

Disadvantages of Finite Element Method

- Large amount of data is required as input for the mesh used in terms of nodal connectivity and other parameters depending on the problem.
- It requires a digital computer and fairly extensive

- It requires longer execution time compared with FEM.
- Output result will vary considerably.

Limitations of FEA

1. Proper engineering judgment is to be exercised to interpret results.
2. It requires large computer memory and computational time to obtain intend results.
3. There are certain categories of problems where other methods are more effective, e.g., fluid problems having boundaries at infinity are better treated by the boundary element method.
4. For some problems, there may be a considerable amount of input data. Errors may creep up in their preparation and the results thus obtained may also appear to be acceptable which indicates deceptive state of affairs. It is always desirable to make a visual check of the input data.
5. In the FEM, many problems lead to round-off errors. Computer works with a limited number of digits and solving the problem with restricted number of digits may not yield the desired degree of accuracy or it may give total erroneous results in some cases. For many problems the increase in the number of digits for the purpose of calculation improves the accuracy.

Applications of FEM

1. **Mechanical engineering**: In mechanical engineering, FEM applications include steady and transient thermal analysis in solids and fluids, stress analysis in solids, automotive design and analysis and manufacturing process simulation.
2. **Geotechnical engineering:** FEM applications include stress analysis, slope stability analysis, soil structure interactions, seepage of fluids in soils and rocks, analysis of dams, tunnels, bore holes, propagation of stress waves and dynamic soil structure interaction.
3. **Aerospace engineering**: FEM is used for several purposes such as structural analysis for natural frequencies, modes shapes, response analysis and aerodynamics.
4. **Nuclear engineering**: FEM applications include steady and dynamic analysis of reactor containment structures, thermo-viscoelastic analysis of reactor components, steady and transient temperature-distribution analysis of reactors and related structures.
5. **Electrical and electronics engineering**: FEM applications include electrical network analysis, electromagnetics, insulation design analysis in high-voltage equipments, dynamic analysis of motors and heat analysis in electrical and electronic equipments.
6. **Metallurgical, chemical engineering**: In metallurgical engineering, FEM is used for the metallurgical process simulation, moulding and casting. In chemical engineering, FEM can be used in the simulation of chemical processes, transport processes and chemical reaction simulations.
7. **Meteorology and bio-engineering**: In the recent times, FEM is used in climate predictions, monsoon prediction and wind predictions. FEM is also used in bio-engineering for the simulation of various human organs, blood circulation prediction and even total synthesis of human body.

8 Civil Engineering Structure: Finite element analysis (FEA) is an extremely useful tool in the field of civil engineering for numerically approximating physical structures that are too complex for regular analytical solutions. Consider a concrete beam with support at both ends, facing a concentrated load on its center span. The deflection at the center span can be determined mathematically in a relatively simple way, as the initial and boundary conditions are finite and in control. However, once you transport the same beam into a practical application, such as within a bridge, the forces at play become much more difficult to analyze with simple mathematics.

finite element method vs classical method

Classical Methods	Finite Element method
1) Exact equations are formed and exact solutions are obtained.	1) Exact equations are formed but approximate solutions are formed.
2) Solutions can be obtained for few standard cases.	2) Solution can be obtained for all problems.
3) For the solution of shape, Boundary conditions and loading some assumptions are made.	3) No assumptions are made problem is treated as it is.
4) When material is not isotropic, solution for the problems becomes very difficult.	4) All type of property can handle without any difficulty.
5) If structure consist more than one material, it is difficult to analyze.	6) If structure consist more than one material then it can be analyzed without any difficulty.

POTENTIAL ENERGY AND EQUILIBRIUM; THE RAYLEIGH–RITZ METHOD

In mechanics of solids, our problem is to determine the displacement **u** of the body shown in Fig. 1.1, satisfying the equilibrium equations 1.6. Note that stresses are related to strains, which, in turn, are related to displacements. This leads to requiring solution of second-order partial differential equations. Solution of this set of equations is generally referred to as an *exact* solution. Such exact solutions are available for simple geometries and loading conditions, and one may refer to publications in theory of elasticity. For problems of complex geometries and general boundary and loading conditions, obtaining such solutions is an almost impossible task. Approximate solution methods usually employ potential energy or variational methods, which place less stringent conditions on the functions.

Potential Energy, Π

The total potential energy Π of an elastic body, is defined as the sum of total strain energy (U) and the work potential:

$$\Pi = \underset{(U)}{\text{Strain energy}} + \underset{(\text{WP})}{\text{Work potential}} \tag{1.24}$$

For linear elastic materials, the strain energy per unit volume in the body is $\frac{1}{2}\boldsymbol{\sigma}^{\mathrm{T}}\boldsymbol{\epsilon}$. For the elastic body shown in Fig. 1.1, the total strain energy U is given by

$$U = \frac{1}{2}\int_V \boldsymbol{\sigma}^{\mathrm{T}}\boldsymbol{\epsilon}\, dV \tag{1.25}$$

The work potential WP is given by

$$\text{WP} = -\int_V \mathbf{u}^{\mathrm{T}}\mathbf{f}\, dV - \int_S \mathbf{u}^{\mathrm{T}}\mathbf{T}\, dS - \sum_i \mathbf{u}_i^{\mathrm{T}}\mathbf{P}_i \tag{1.26}$$

The total potential for the general elastic body shown in Fig. 1.1 is

Principle of Minimum Potential Energy

For conservative systems, of all the kinematically admissible displacement fields, those corresponding to equilibrium extremize the total potential energy. If the extremum condition is a minimum, the equilibrium state is stable.

Example 1.2

The potential energy for the linear elastic one-dimensional rod (Fig. E1.2), with body forc neglected, is

$$\Pi = \frac{1}{2}\int_0^L EA\left(\frac{du}{dx}\right)^2 dx - 2u_1$$

where $u_1 = u(x = 1)$.

Let us consider a polynomial function

$$u = a_1 + a_2 x + a_3 x^2$$

This must satisfy $u = 0$ at $x = 0$ and $u = 0$ at $x = 2$. Thus,

$$0 = a_1$$
$$0 = a_1 + 2a_2 + 4a_3$$

Hence,

$$a_2 = -2a_3$$
$$u = a_3(-2x + x^2) \qquad u_1 = -a_3$$

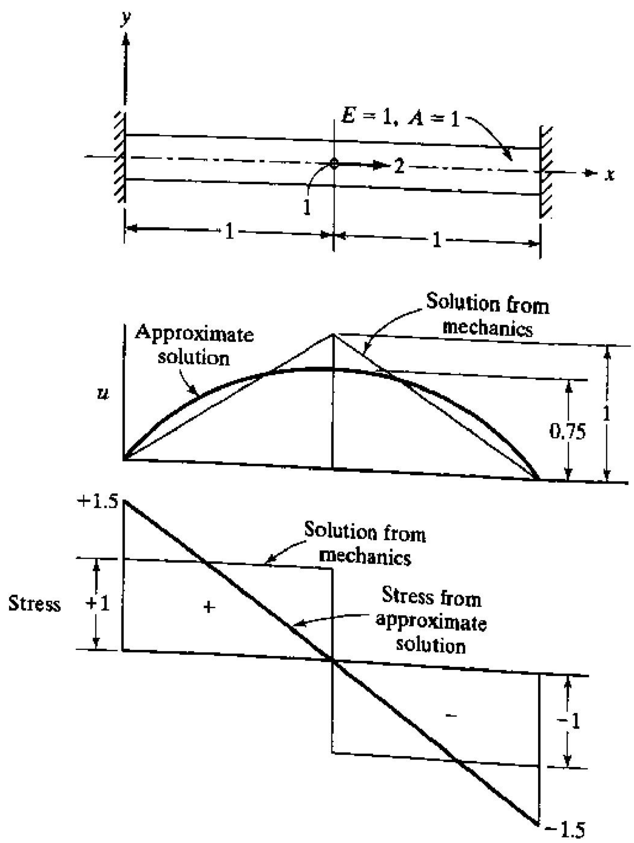


FIGURE E1.2

Then $du/dx = 2a_3(-1 + x)$ and

$$\Pi = \frac{1}{2}\int_0^2 4a_3^2(-1 + x)^2\,dx - 2(-a_3)$$

$$= 2a_3^2\int_0^2 (1 - 2x + x^2)\,dx + 2a_3$$

$$= 2a_3^2\left(\tfrac{2}{3}\right) + 2a_3$$

We set $\partial\Pi/\partial a_3 = 4a_3\left(\frac{2}{3}\right) + 2 = 0$, resulting in

$$a_3 = -0.75 \qquad u_1 = -a_3 = 0.75$$

The stress in the bar is given by

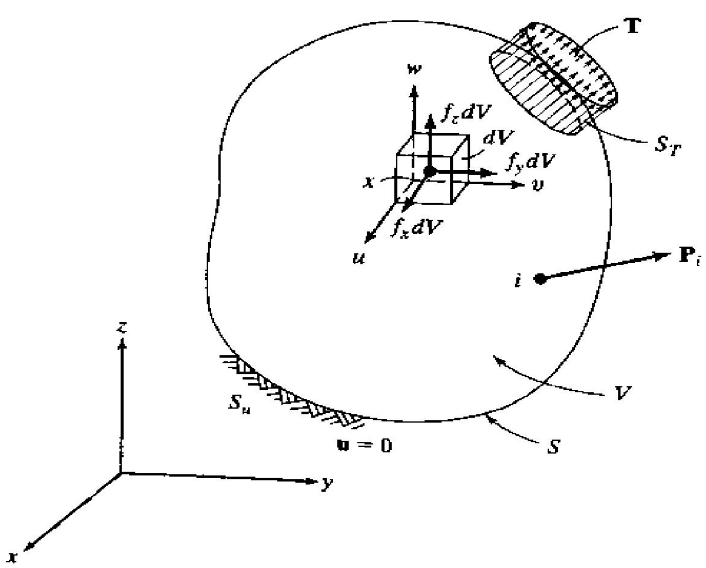
$$\sigma = E\frac{du}{dx} = 1.5(1 - x)$$ ■

We note here that an exact solution is obtained if piecewise polynomial interpolation is used in the construction of u.

The finite element method provides a systematic way of constructing the basis functions ϕ_i used in Eq. 1.30.

STRESSES AND EQUILIBRIUM

A three-dimensional body occupying a volume V and having a surface S is shown in Fig. 1.1. Points in the body are located by x, y, z coordinates. The boundary is constrained on some region, where displacement is specified. On part of the boundary, dis-



tributed force per unit area **T**, also called traction, is applied. Under the force, the body deforms. The deformation of a point **x** ($= [x, y, z]^{\mathrm{T}}$) is given by the three components of its displacement:

$$\mathbf{u} = [u, v, w]^{\mathrm{T}} \tag{1.1}$$

The distributed force per unit volume, for example, the weight per unit volume, is the vector **f** given by

$$\mathbf{f} = [f_x, f_y, f_z]^{\mathrm{T}} \tag{1.2}$$

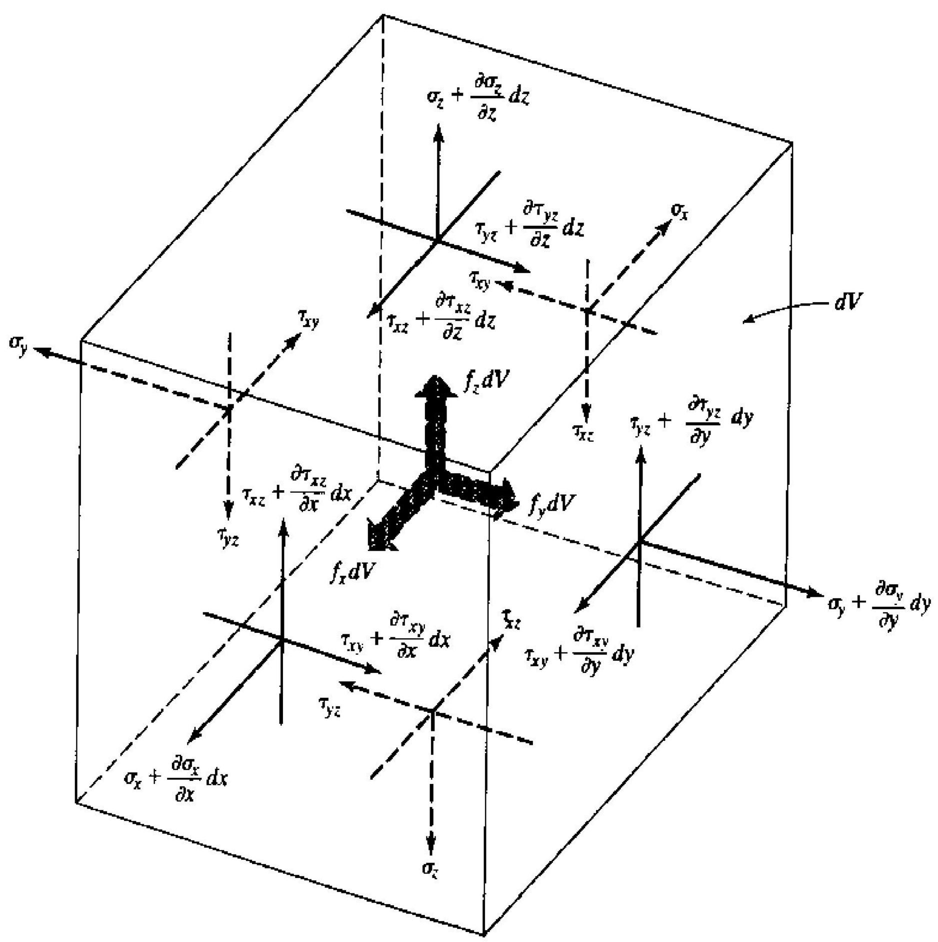
The body force acting on the elemental volume dV is shown in Fig. 1.1. The surface traction **T** may be given by its component values at points on the surface:

$$\mathbf{T} = [T_x, T_y, T_z]^{\mathrm{T}} \tag{1.3}$$

Examples of traction are distributed contact force and action of pressure. A load **P** acting at a point i is represented by its three components:

$$\mathbf{P}_i = [P_x, P_y, P_z]_i^{\mathrm{T}} \tag{1.4}$$

The stresses acting on the elemental volume dV are shown in Fig. 1.2. When the volume dV shrinks to a point, the stress tensor is represented by placing its components in a



(3 × 3) symmetric matrix. However, we represent stress by the six independent components as in

$$\boldsymbol{\sigma} = [\sigma_x, \sigma_y, \sigma_z, \tau_{yz}, \tau_{xz}, \tau_{xy}]^{\mathrm{T}} \tag{1.5}$$

where σ_x, σ_y, σ_z are normal stresses and τ_{yz}, τ_{xz}, τ_{xy}, are shear stresses. Let us consider equilibrium of the elemental volume shown in Fig. 1.2. First we get forces on faces by multiplying the stresses by the corresponding areas. Writing $\Sigma F_x = 0$, $\Sigma F_y = 0$, and $\Sigma F_z = 0$ and recognizing $dV = dx\,dy\,dz$, we get the equilibrium equations

$$\begin{aligned}
\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + f_x &= 0 \\
\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + f_y &= 0 \\
\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + f_z &= 0
\end{aligned} \tag{1.6}$$

Special Cases

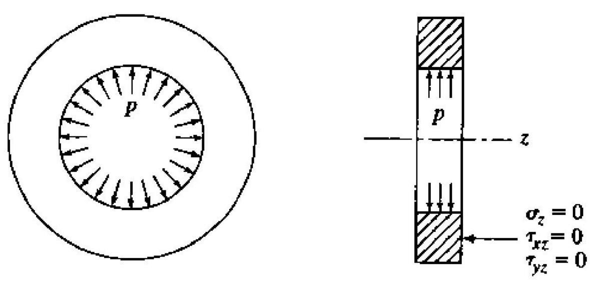
One dimension. In one dimension, we have normal stress σ along x and the corresponding normal strain ϵ. Stress–strain relations (Eq. 1.14) are simply

$$\sigma = E\epsilon \tag{1.16}$$

Two dimensions. In two dimensions, the problems are modeled as plane stress and plane strain.

Plane Stress. A thin planar body subjected to in-plane loading on its edge surface is said to be in plane stress. A ring press fitted on a shaft, Fig. 1.5a, is an example. Here stresses σ_z, τ_{xz}, and τ_{yz} are set as zero. The Hooke's law relations (Eq. 1.11) then give us

$$\begin{aligned}
\epsilon_x &= \frac{\sigma_x}{E} - \nu\frac{\sigma_y}{E} \\
\epsilon_y &= -\nu\frac{\sigma_x}{E} + \frac{\sigma_y}{E} \\
\gamma_{xy} &= \frac{2(1+\nu)}{E}\tau_{xy} \\
\epsilon_z &= -\frac{\nu}{E}(\sigma_x + \sigma_y)
\end{aligned} \tag{1.17}$$



(a)

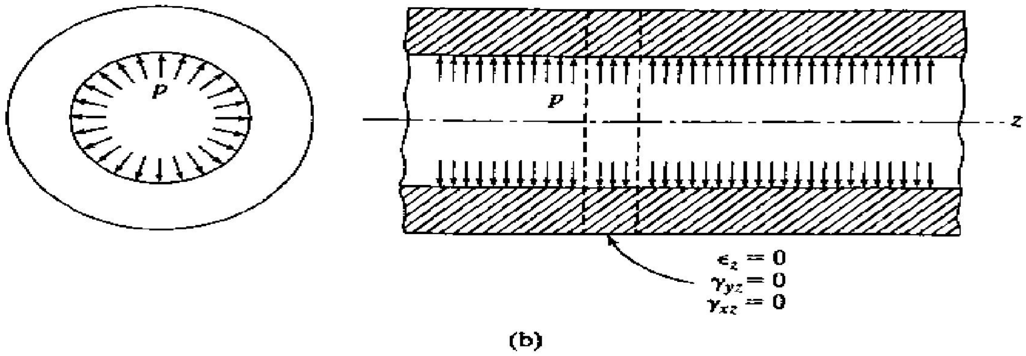


FIGURE 1.5 (a) Plane stress and (b) plane strain.

The inverse relations are given by

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix}\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} \tag{1.18}$$

which is used as $\boldsymbol{\sigma} = \mathbf{D}\boldsymbol{\epsilon}$.

Plane Strain. If a long body of uniform cross section is subjected to transverse loading along its length, a small thickness in the loaded area, as shown in Fig. 1.5b, can be treated as subjected to plane strain. Here ϵ_z, γ_{zx}, γ_{yz} are taken as zero. Stress σ_z may not be zero in this case. The stress–strain relations can be obtained directly from Eqs. 1.14 and 1.15:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1}{2}-\nu \end{bmatrix}\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} \tag{1.19}$$

D here is a (3×3) matrix, which relates three stresses and three strains.

Anisotropic bodies, with uniform orientation, can be considered by using the appropriate **D** matrix for the material.

Strain Displacement Relationship for Axisymmentric element:

Consider an axisymmetric ring element and its cross section to represent the general state of strain for an axisymmetric problem. The displacements can be expressed for element *ABCD* in the plane of a cross-section in cylindrical coordinates. We then let *u* and *w* denote the displacements in the radial and longitudinal directions, respectively. The side *AB* of the element is displaced an amount *u,* and side *CD* is then displaced an amount *u* + in the radial direction.

The strain in the tangential direction depends on the tangential displacement v and on the radial displacement u.
However, for axisymmetric deformation behavior, recall that the tangential displacement v is equal to zero.

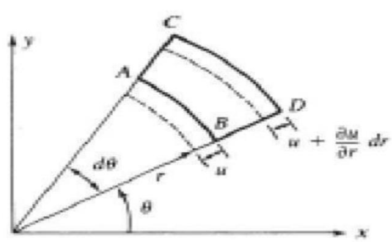
The tangential strain is due only to the radial displacement.

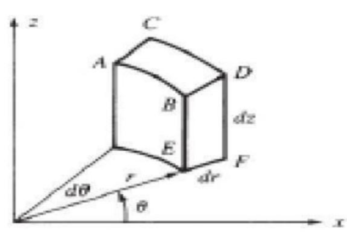
Having only radial displacement *u*, the new length of the arc *AB* is $(r + u)d\theta$, and the tangential strain is then given by:

$$\varepsilon_\theta = \frac{(r+u)d\theta - rd\theta}{rd\theta} = \frac{u}{r}$$

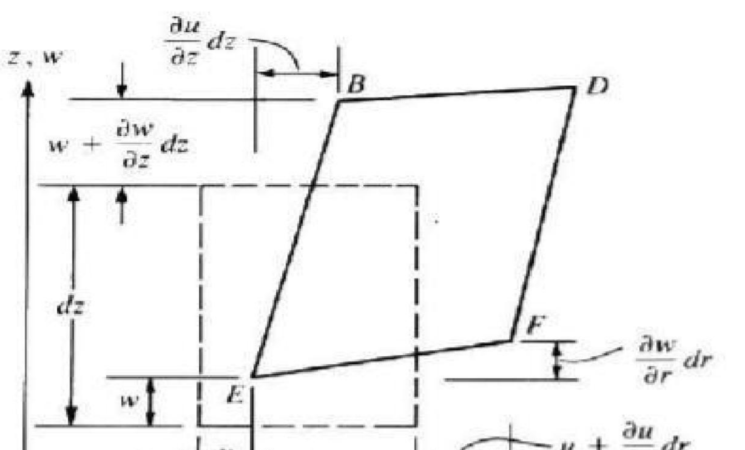
Consider the longitudinal element BDEF to obtain the longitudinal strain and the shear strain. The element displaces by amounts u and w in the radial and longitudinal directions at point E.

The element displaces additional amounts:
$(\partial w/\partial z)dz$ along line *BE* and
$(\partial u/\partial r)dr$ along line *EF*.





The normal strain in the radial direction is then given by: $\varepsilon_r = \frac{\partial u}{\partial r}$



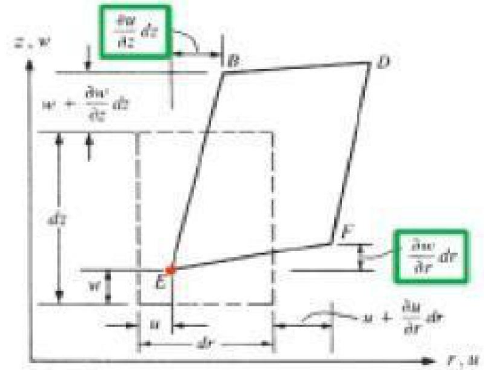
Furthermore, observing lines *EF* and *BE*, we see that point F moves upward an amount $(\partial w/\partial r)dr$ with respect to point *E* and point *B* moves to the right an amount $(\partial u/\partial z)dz$ with respect to point *E*.

The longitudinal normal strain is given by:

$$\varepsilon_z = \frac{\partial w}{\partial z}$$

The shear strain in the *r-z* plane is:

$$\gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$



Summarizing the strain-displacement relationships gives:

$$\varepsilon_r = \frac{\partial u}{\partial r} \qquad \varepsilon_\theta = \frac{u}{r} \qquad \varepsilon_z = \frac{\partial w}{\partial z} \qquad \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$