Lecture 1: Introduction

1.1.1 Introduction

The Finite Element Method (FEM) is a numerical technique to find approximate solutions of partial differential equations. It was originated from the need of solving complex elasticity and structural analysis problems in Civil, Mechanical and Aerospace engineering. In a structural simulation, FEM helps in producing stiffness and strength visualizations. It also helps to minimize material weight and its cost of the structures. FEM allows for detailed visualization and indicates the distribution of stresses and strains inside the body of a structure. Many of FE software are powerful yet complex tool meant for professional engineers with the training and education necessary to properly interpret the results.

Several modern FEM packages include specific components such as fluid, thermal, electromagnetic and structural working environments. FEM allows entire designs to be constructed, refined and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The use of FEM has significantly decreased the time to take products from concept to the production line. One must take the advantage of the advent of faster generation of personal computers for the analysis and design of engineering product with precision level of accuracy.

1.1.2 Background of Finite Element Analysis

The finite element analysis can be traced back to the work by Alexander Hrennikoff (1941) and Richard Courant (1942). Hrenikoff introduced the framework method, in which a plane elastic medium was represented as collections of bars and beams. These pioneers share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

- In 1950s, solution of large number of simultaneous equations became possible because of the digital computer.
- In 1960, Ray W. Clough first published a paper using term “Finite Element Method”.
- In 1965, First conference on “finite elements” was held.
- In 1967, the first book on the “Finite Element Method” was published by Zienkiewicz and Chung.
- In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems.

- In the 1970s, most commercial FEM software packages (ABAQUS, NASTRAN, ANSYS, etc.) originated. Interactive FE programs on supercomputer lead to rapid growth of CAD systems.
- In the 1980s, algorithm on electromagnetic applications, fluid flow and thermal analysis were developed with the use of FE program.
- Engineers can evaluate ways to control the vibrations and extend the use of flexible, deployable structures in space using FE and other methods in the 1990s. Trends to solve fully coupled solution of fluid flows with structural interactions, bio-mechanics related problems with a higher level of accuracy were observed in this decade.

With the development of finite element method, together with tremendous increases in computing power and convenience, today it is possible to understand structural behavior with levels of accuracy. This was in fact the beyond of imagination before the computer age.

1.1.3 Numerical Methods

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature. For numerical approach, the solutions will be approximate when any of these relations are only approximately satisfied. The numerical method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. Few numerical methods which are commonly used to solve solid and fluid mechanics problems are given below.

- Finite Difference Method
- Finite Volume Method
- Finite Element Method
- Boundary Element Method
- Meshless Method

The application of finite difference method for engineering problems involves replacing the governing differential equations and the boundary condition by suitable algebraic equations. For example in the analysis of beam bending problem the differential equation is reduced to be solution of algebraic equations written at every nodal point within the beam member. For example, the beam equation can be expressed as:

$$\frac{d^4 w}{dx^4} = \frac{q}{EI} \qquad (1.1.1)$$

To explain the concept of finite difference method let us consider a displacement function variable namely $w = f(x)$

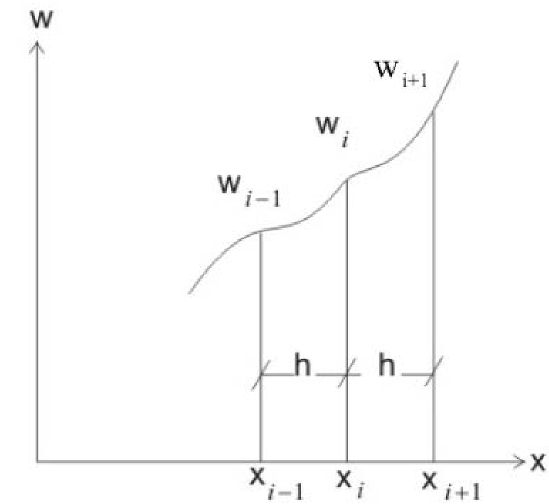


Fig. 1.1.1 Displacement Function

Now, $\Delta w = f(x + \Delta x) - f(x)$

So, $$\frac{dw}{dx} = \frac{\Delta w}{\Delta x} = \underset{\Delta x \to 0}{\mathrm{Lt}} \frac{\Delta w}{\Delta x} = \underset{\Delta x \to 0}{\mathrm{Lt}} \frac{f(x+\Delta x) - f(x)}{\Delta x} = \frac{1}{h}\left(w_{i+1} - w_i\right) \qquad (1.1.2)$$

Thus,

$$\frac{d^2 w}{dx^2} = \frac{d}{dx}\left[\frac{1}{h}\left(w_{i+1} - w_i\right)\right] = \frac{1}{h^2}\left(w_{i+2} - w_{i+1} - w_{i+1} + w_i\right) = \frac{1}{h^2}\left(w_{i+2} - 2\,w_{i+1} + w_i\right) \qquad (1.1.3)$$

$$\frac{d^3 w}{dx^3} = \frac{1}{h^3}\left(w_{i+3} - w_{i+2} - 2w_{i+2} + 2w_{i+1} + w_{i+1} - w_i\right)$$
$$= \frac{1}{h^3}\left(w_{i+3} - 3w_{i+2} + 3w_{i+1} - w_i\right) \qquad (1.1.4)$$

$$\frac{d^4w}{dx^4} = \frac{1}{h^4}\left(w_{i+4} - w_{i+3} - 3w_{i+3} + 3w_{i+2} + 3w_{i+2} - 3w_{i+1} - w_{i+1} + w_i\right)$$
$$= \frac{1}{h^4}\left(w_{i+4} - 4w_{i+3} + 6w_{i+2} - 4w_{i+1} + w_i\right) \qquad (1.1.5)$$
$$= \frac{1}{h^4}\left(w_{i+2} - 4w_{i+1} + 6w_i - 4w_{i-1} + w_{i-2}\right)$$

Thus, eq. (1.1.1) can be expressed with the help of eq. (1.1.5) and can be written in finite difference form as:

$$(w_{i-2} - 4w_{i-1} + 6w_i - 4w_{i+1} + w_{i+2}) = \frac{q}{EI}h^4 \qquad (1.1.6)$$

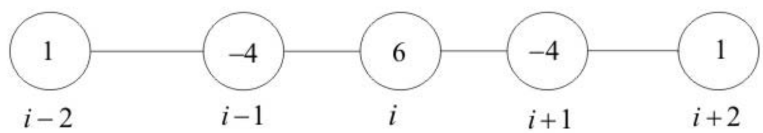


Fig. 1.1.2 Finite difference equation at node i

Thus, the displacement at node *i* of the beam member corresponds to uniformly distributed load can be obtained from eq. (1.1.6) with the help of boundary conditions. It may be interesting to note that, the concept of node is used in the finite difference method. Basically, this method has an array of grid points and is a point wise approximation, whereas, finite element method has an array of small interconnecting sub-regions and is a piece wise approximation.

Each method has noteworthy advantages as well as limitations. However it is possible to solve various problems by finite element method, even with highly complex geometry and loading conditions, with the restriction that there is always some numerical errors. Therefore, effective and reliable use of this method requires a solid understanding of its limitations.

1.1.4 Concepts of Elements and Nodes

Any continuum/domain can be divided into a number of pieces with very small dimensions. These small pieces of finite dimension are called 'Finite Elements' (Fig. 1.1.3). A field quantity in each element is allowed to have a simple spatial variation which can be described by polynomial terms. Thus the original domain is considered as an assemblage of number of such small elements. These elements are connected through number of joints which are called 'Nodes'. While discretizing the structural system, it is assumed that the elements are attached to the adjacent elements only at the nodal points. Each element contains the material and geometrical properties. The material properties inside an element are assumed to be constant. The elements may be 1D elements, 2D elements or 3D elements. The physical object can be modeled by choosing appropriate element such as frame element, plate element, shell element, solid element, etc. All elements are then assembled to obtain the solution of the entire domain/structure under certain loading conditions. Nodes are assigned at a certain density throughout the continuum depending on the anticipated stress levels of a particular domain. Regions which will receive large amounts of stress variation usually have a higher node density than those which experience little or no stress.

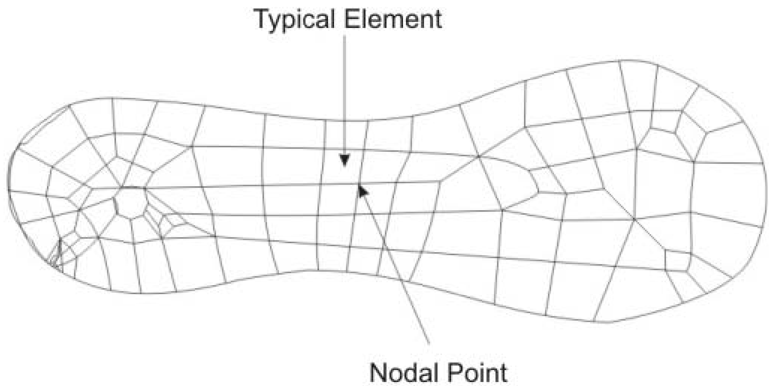


Fig. 1.1.3 Finite element discretization of a domain

1.1.5 Degrees of Freedom

A structure can have infinite number of displacements. Approximation with a reasonable level of accuracy can be achieved by assuming a limited number of displacements. This finite number of displacements is the number of degrees of freedom of the structure. For example, the truss member will undergo only axial deformation. Therefore, the degrees of freedom of a truss member with respect to its own coordinate system will be one at each node. If a two dimension structure is modeled by truss elements, then the deformation with respect to structural coordinate system will be two and therefore degrees of freedom will also become two. The degrees of freedom for various types of element are shown in Fig. 1.1.4 for easy understanding. Here (u,v,w) and $(\theta_x,\theta_y,\theta_z)$ represent displacement and rotation respectively.

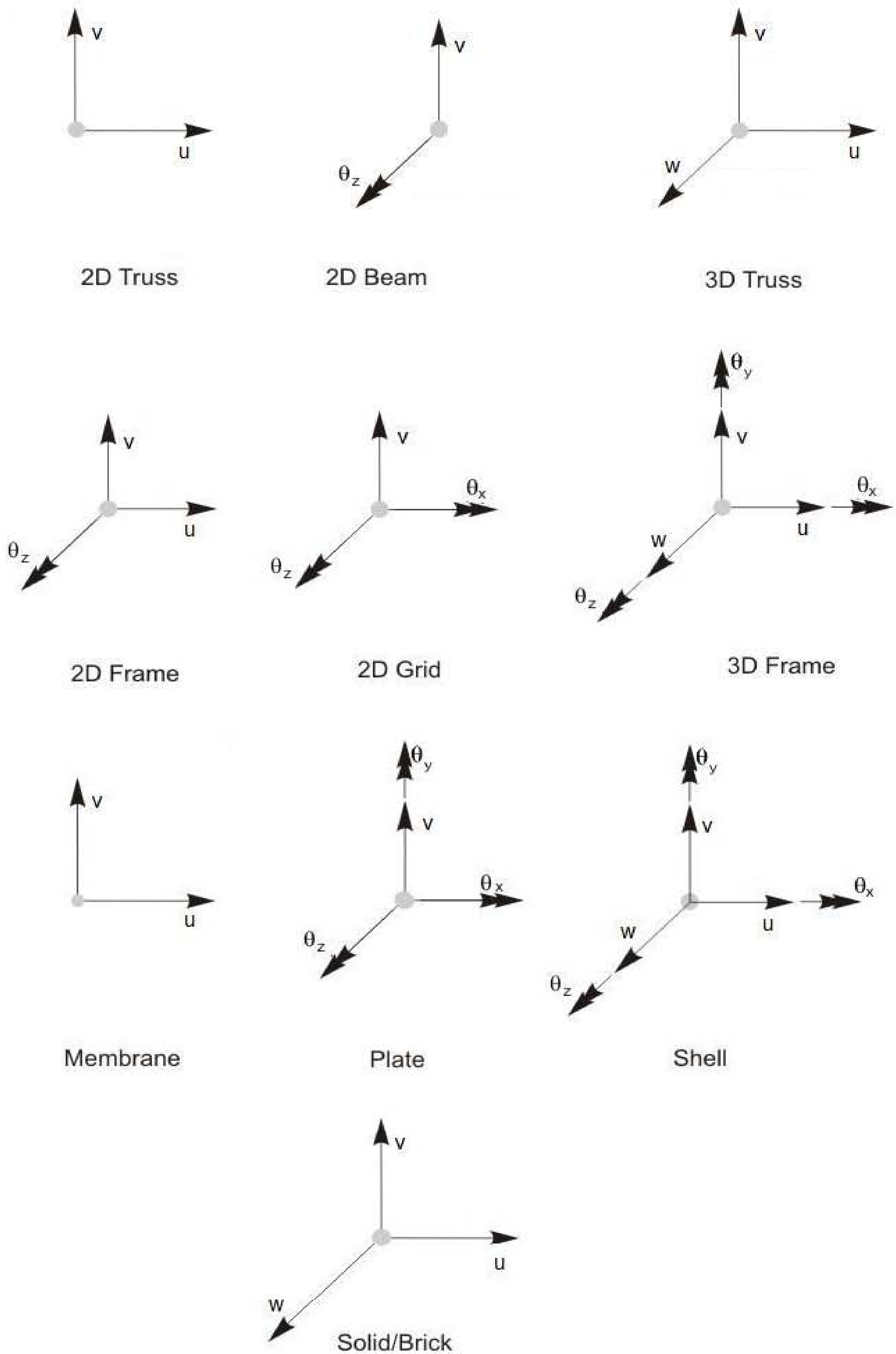


Fig. 1.1.4 Degrees of Freedom for Various Elements

Lecture 2: Basic Concepts of Finite Element Analysis

1.2.1 Idealization of a Continuum

A continuum may be discretized in different ways depending upon the geometrical configuration of the domain. Fig. 1.2.1 shows the various ways of idealizing a continuum based on the geometry.

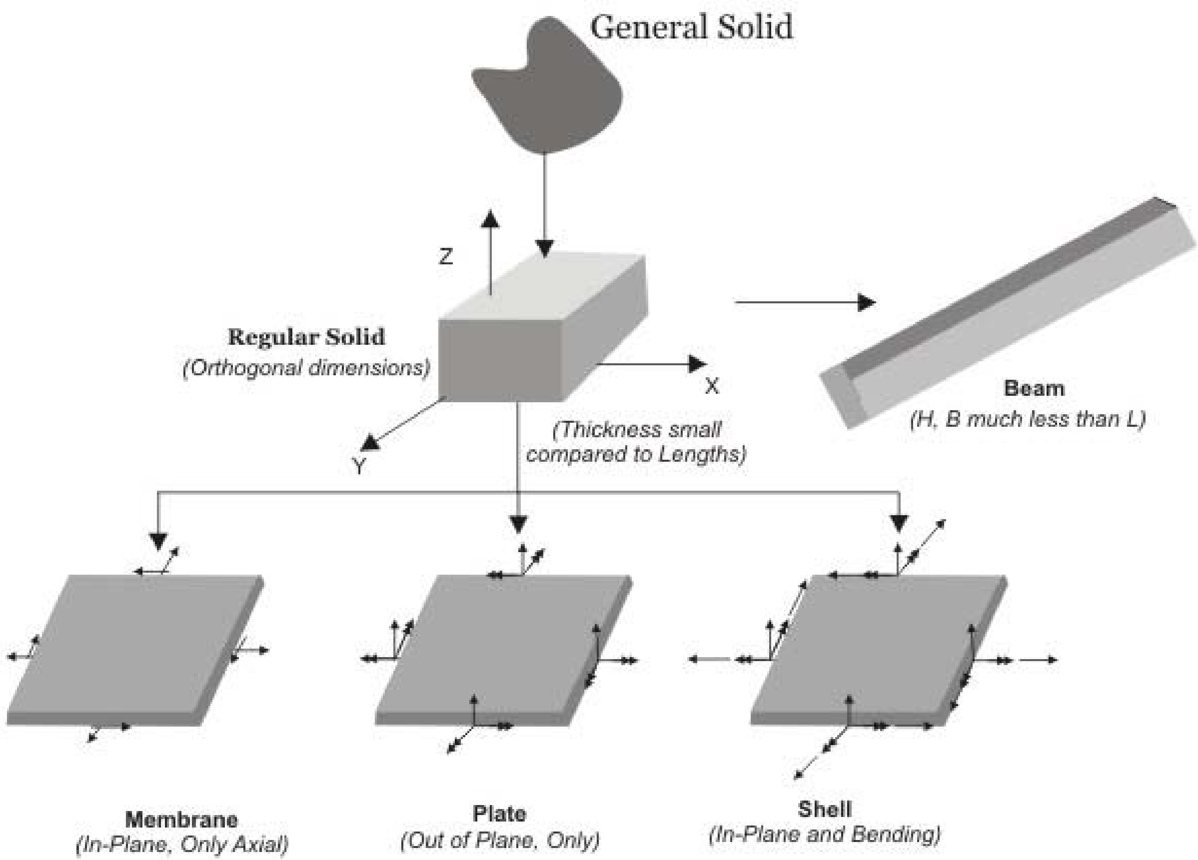


Fig. 1.2.1 Various ways of Idealization of a Continuum

1.2.2 Discretization of Technique

The need of finite element analysis arises when the structural system in terms of its either geometry, material properties, boundary conditions or loadings is complex in nature. For such case, the whole structure needs to be subdivided into smaller elements. The whole structure is then analyzed by the assemblage of all elements representing the complete structure including its all properties.

The subdivision process is an important task in finite element analysis and requires some skill and knowledge. In this procedure, first, the number, shape, size and configuration of elements have to be decided in such a manner that the real structure is simulated as closely as possible. The discretization is to be in such that the results converge to the true solution. However, too fine mesh will lead to extra computational effort. Fig. 1.2.2 shows a finite element mesh of a continuum using triangular and quadrilateral elements. The assemblage of triangular elements in this case shows better representation of the continuum. The discretization process also shows that the more accurate representation is possible if the body is further subdivided into some finer mesh.

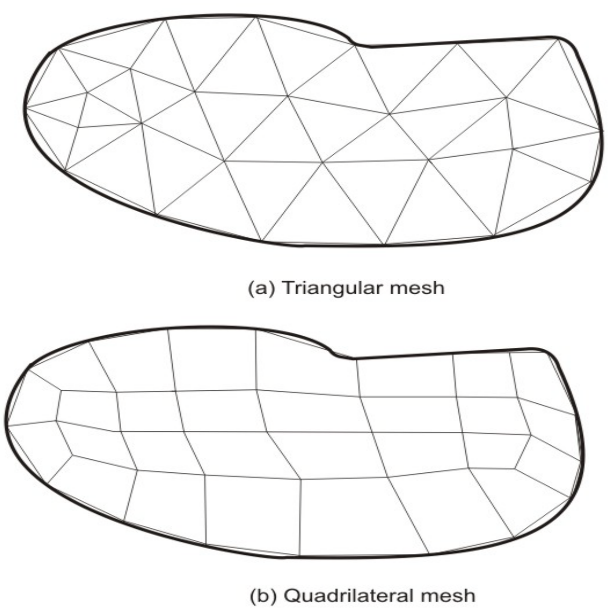


Fig. 1.2.2 Discretization of a continuum

1.2.3 Concepts of Finite Element Analysis

FEA consists of a computer model of a continuum that is stressed and analyzed for specific results. A continuum has infinite particles with continuous variation of material properties. Therefore, it needs to simplify to a finite size and is made up of an assemblage of substructures, components and members. Discretization process is necessary to convert whole structure to an assemblage of members/elements for determining its responses. Fig. 1.2.3 shows the process of idealization of actual structure to a finite element form to obtain the response results. The assumptions are required to be made by the experienced engineer with finite element background for getting appropriate response results. On the basis of assumptions, the appropriate constitutive model can be constructed. For the linear-elastic-static analysis of structures, the final form of equation will be made in the form of $F=Kd$ where F, K and d are the nodal loads, global stiffness and nodal displacements respectively.

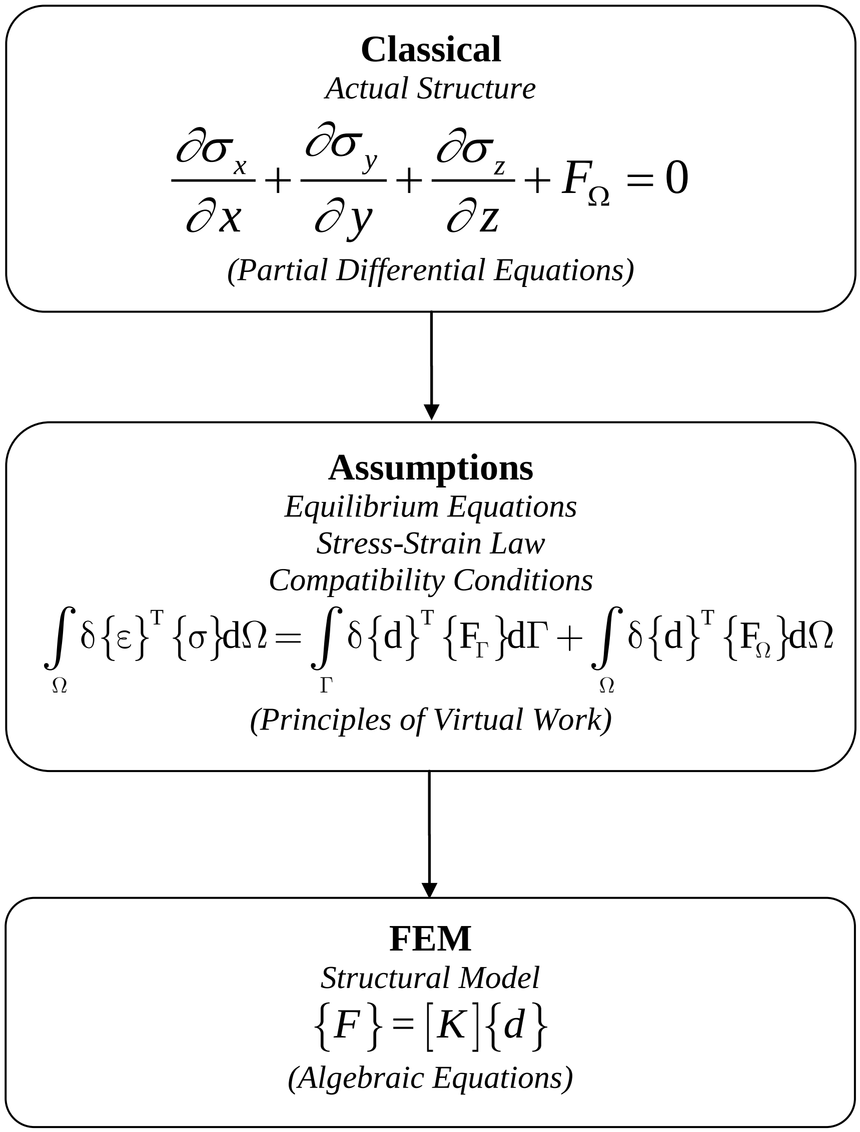


Fig. 1.2.3From classical to FE solution

Varieties of engineering problem like solid and fluid mechanics, heat transfer can easily be solved by the concept of finite element technique. The basic form of the equation will become as follows where action, property and response parameter will vary for case to case as outlined in Table 1.2.1.

$$\{F\} = [K]\{d\} \quad OR \quad \{d\} = [K]^{-1}\{F\}$$

Action Property Response

Table 1.2.1 Response parameter for different cases

	Property	**Action**	**Response**
Solid	Stiffness	Load	Displacement
Fluid	Viscosity	Body force	Pressure/Velocity
Thermal	Conductivity	Heat	Temperature

1.2.4 Advantages of FEA

1. The physical properties, which are intractable and complex for any closed bound solution, can be analyzed by this method.
2. It can take care of any geometry (may be regular or irregular).
3. It can take care of any boundary conditions.
4. Material anisotropy and non-homogeneity can be catered without much difficulty.
5. It can take care of any type of loading conditions.
6. This method is superior to other approximate methods like Galerkine and Rayleigh-Ritz methods.
7. In this method approximations are confined to small sub domains.
8. In this method, the admissible functions are valid over the simple domain and have nothing to do with boundary, however simple or complex it may be.
9. Enable to computer programming.

1.2.5 Disadvantages of FEA

1. Computational time involved in the solution of the problem is high.
2. For fluid dynamics problems some other methods of analysis may prove efficient than the FEM.

1.2.6 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.

1.2.7 Errors and Accuracy in FEA

Every physical problem is formulated by simplifying certain assumptions. Solution to the problem, classical or numerical, is to be viewed within the constraints imposed by these simplifications. The material may be assumed to be homogeneous and isotropic; its behavior may be considered as linearly elastic; the prediction of the exact load in any type of structure is next to impossible. As such the true behavior of the structure is to be viewed with in these constraints and obvious errors creep in engineering calculations.

1. The results will be erroneous if any mistake occurs in the input data. As such, preparation of the input data should be made with great care.
2. When a continuum is discretised, an infinite degrees of freedom system is converted into a model having finite number of degrees of freedom. In a continuum, functions which are continuous are now replaced by ones which are piece-wise continuous within individual elements. Thus the actual continuum is represented by a set of approximations.
3. The accuracy depends to a great extent on the mesh grading of the continuum. In regions of high strain gradient, higher mesh grading is needed whereas in the regions of lower strain, the mesh chosen may be coarser. As the element size decreases, the discretisation error reduces.
4. Improper selection of shape of the element will lead to a considerable error in the solution. Triangle elements in the shape of an equilateral or rectangular element in the shape of a square will always perform better than those having unequal lengths of the sides. For very long shapes, the attainment of convergence is extremely slow.
5. In the finite element analysis, the boundary conditions are imposed at the nodes of the element whereas in an actual continuum, they are defined at the boundaries. Between the nodes, the actual boundary conditions will depend on the shape functions of the element forming the boundary.

6. Simplification of the boundary is another source of error. The domain may be reduced to the shape of polygon. If the mesh is refined, then the error involved in the discretised boundary may be reduced.
7. During arithmetic operations, the numbers would be constantly round-off to some fixed working length. These round–off errors may go on accumulating and then resulting accuracy of the solution may be greatly impaired.

Lecture 3: Introduction to Elasticity

1.3.1 Stresses and Equilibrium

Let consider an infinitesimal element of sides dx, dy and dz as shown in Fig. 1.3.1. The stresses are acting on the elemental volume dV because of external and or body forces. These stresses can be represented by six independent components as given below.

$$\{\sigma\} = \left[\sigma_x, \quad \sigma_y, \quad \sigma_z, \quad \tau_{xy}, \quad \tau_{yz}, \quad \tau_{zx}\right]^T \tag{1.3.1}$$

Here, σ_x, σ_y *and* σ_z are normal stresses and τ_{xy}, τ_{yz} *and* τ_{zx} are shear stresses. Applying the conditions of static equilibrium for forces along the direction of X axis (i.e., $\sum F_x = 0$), following expression will be obtained.

$$\frac{\partial \sigma_x}{\partial x} dxdydz + \frac{\partial \tau_{yx}}{\partial y} dxdydz + \frac{\partial \tau_{zx}}{\partial z} dxdydz + F_{\Omega x} dxdydz = 0 \tag{1.3.2}$$

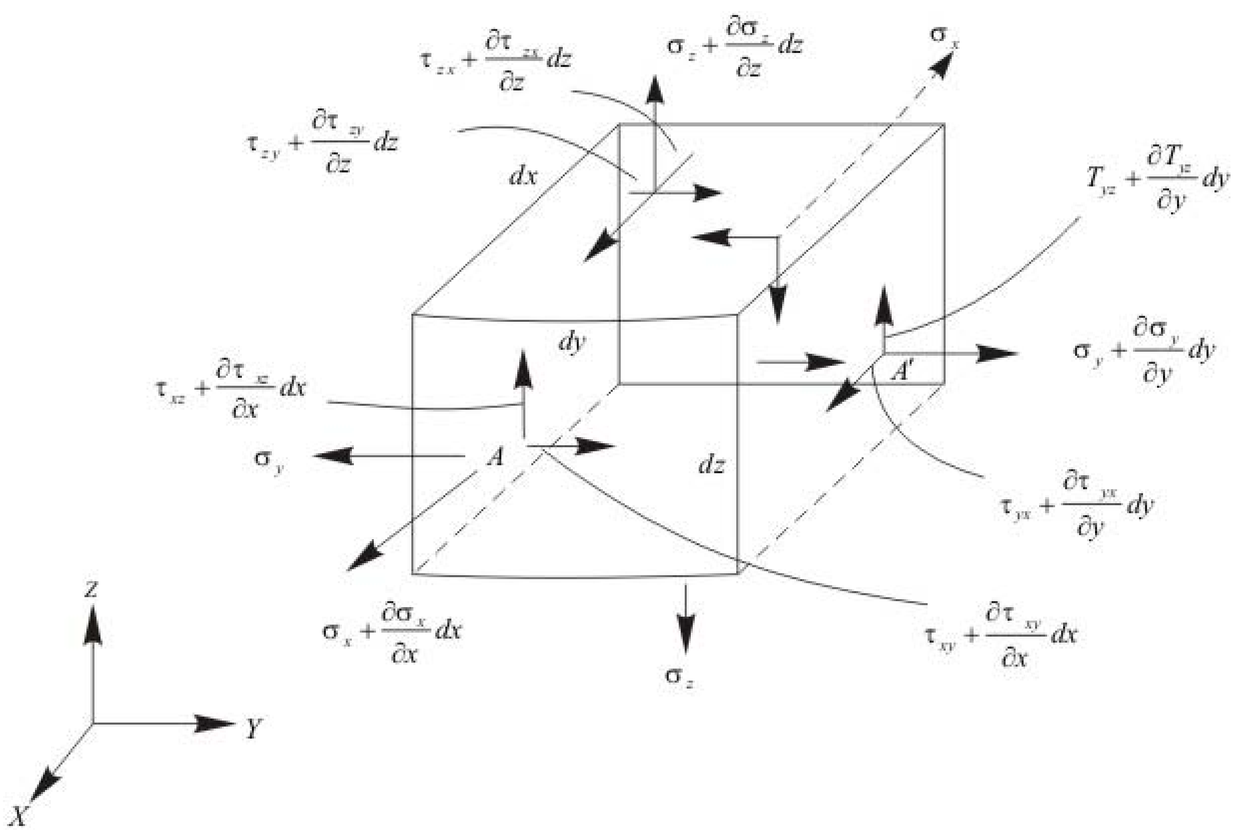


Fig. 1.3.1Stresses on an infinitesimal element

Where, $F_{\Omega x}$ is the component of body force along x direction. Now, dividing dxdydz on the above expression, following equilibrium condition is obtained.

$$\frac{\partial \sigma_x}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}=-f_x \qquad (1.3.3)$$

Similarly, applying equilibrium condition along *Y* and *Z* directions, one can find the following relations.

$$\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \sigma_y}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}=-f_y \qquad (1.3.4)$$

$$\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \sigma_z}{\partial z}=-f_z \qquad (1.3.5)$$

Here, $F_{\Omega y}$ and $F_{\Omega z}$ are the component of body forces along *Y* and *Z* directions respectively. Satisfying moment equations (i.e., $\sum M_x = 0; \sum M_y = 0 \ and \ \sum M_z = 0;$), one can obtain the following relations.

$$\tau_{xy}=\tau_{yx}; \quad \tau_{yz}=\tau_{zy} \text{ and } \tau_{xz}=\tau_{zx} \qquad (1.3.6)$$

Using eq. (1.3.6), the equilibrium equations (1.3.3 to 1.3.5), can be rewritten in the following form.

$$\begin{aligned}
&\frac{\partial \sigma_x}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}=-F_{\Omega x}\\
&\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \sigma_y}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}=-F_{\Omega y}\\
&\frac{\partial \tau_{zx}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \sigma_z}{\partial z}=-F_{\Omega z}
\end{aligned} \qquad (1.3.7)$$

Eq. (1.3.7) is known as equation of equilibrium.

Let assume an element of area $\Delta\Gamma$ on the surface of the solid in equilibrium (Fig.1.3.2) and $F_{\Gamma x}$, $F_{\Gamma y}$ and $F_{\Gamma z}$ are the components of external forces per unit area and are acting on the surface. Consideration of equilibrium along the three axes directions gives the following relations.

$$\begin{aligned}
&\sigma_x l+\tau_{xy} m+\tau_{zx} n=F_{\Gamma x}\\
&\tau_{xy} l+\sigma_y m+\tau_{yz} n=F_{\Gamma y}\\
&\tau_{zx} l+\tau_{yz} m+\sigma_z n=F_{\Gamma z}
\end{aligned} \qquad (1.3.8)$$

Here, *l*, *m* and *n* are the direction cosines of the normal to the boundary surface. Eq. (1.3.8) is known as **static boundary condition**.

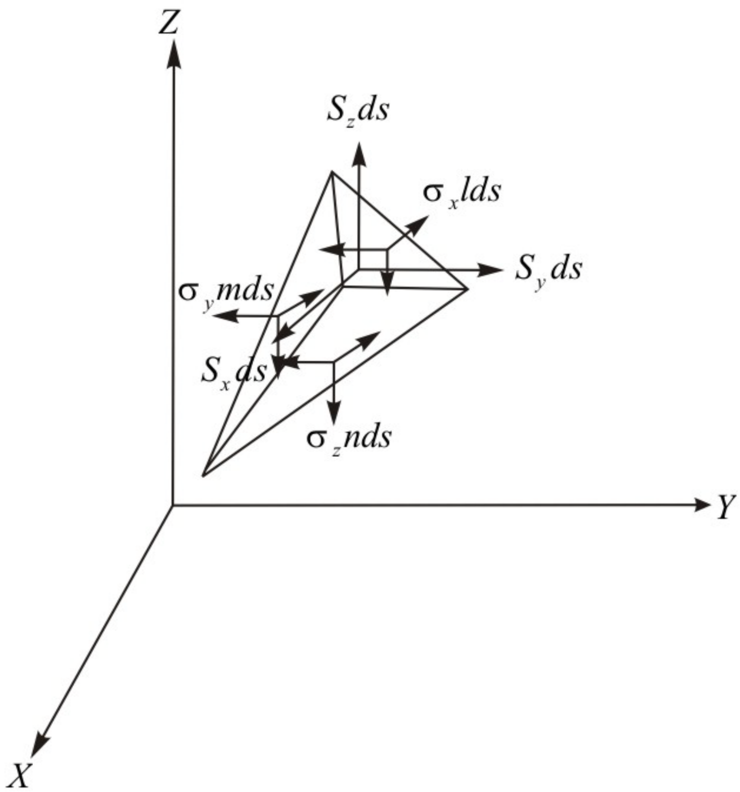


Fig. 1.3.2 Forces acting on an elemnt on the boundary

1.3.2 Strain-Displacement Relations

The displacement at any point of a deformable body may be expressed by the components of *u*, *v* and *w* parallel to the Cartesian coordinate's axes. The components of the displacements can be described as functions of *x*, *y* and *z*. Displacements basically the change of position during deformation. If point P (x,y,z) is displaced to P' (x',y',z'), then the displacement along X, Y and Z direction (Fig. 1.3.3) will become

x' = x + u or u = x' – x
y' = y + v or v = y' – y
z' = z + w or w = z' – z

Therefore, the normal strain can be written as:

$$\varepsilon_x = \underset{\Delta x \to 0}{Lt} \frac{\Delta u}{\Delta x} = \frac{\partial u}{\partial x} \text{ (As } \varepsilon = \frac{\Delta L}{L} \text{ for uniform strain in axial member)}$$

Similarly, $\varepsilon_y = \frac{\partial v}{\partial y}$ and $\varepsilon_z = \frac{\partial w}{\partial z}$

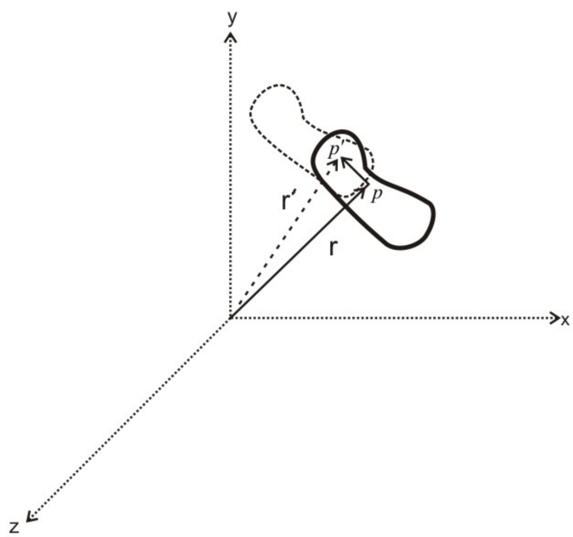


Fig. 1.3.3 Deformation of an elastic body

Let consider points P, Q and R are before deformation and points P’,Q’ and R’ are after deformation as shown in Fig. 1.3.4 below. Now for small deformation, rotation of PQ will become

$$\theta_1 = \lim_{\Delta x \to 0} \frac{\Delta v}{\Delta x} = \frac{\partial v}{\partial x}$$

Similarly, rotation of PR due to deformation will be: $\theta_2 = \lim_{\Delta y \to 0} \frac{\Delta u}{\Delta y} = \frac{\partial u}{\partial y}$

Thus, the total change of angle between PQ and PR after deformation is as follows which is defined as shear strain in X-Y plane.

$$\gamma_{xy} = \gamma_{yx} = \theta_1 + \theta_2 = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

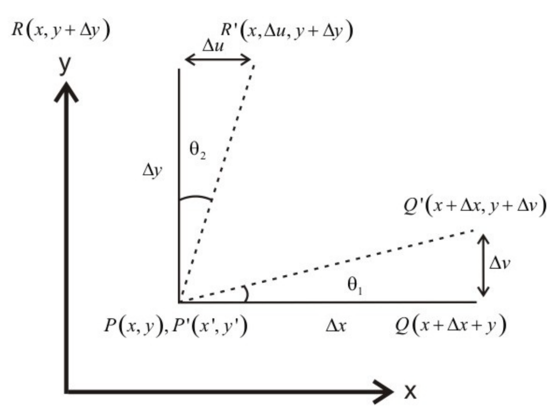


Fig. 1.3.4 Derivation of shear strain

Similarly, shear strains in Y-Z and X-Z plane will become

$$\gamma_{yz} = \gamma_{zy} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}; \ \gamma_{xz} = \gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$

The strain can be expressed as partial derivatives of the displacements *u*, *v* and *w*. The above expressions for strain-displacement relationship are true only for small amplitude of deformation. However, the strain-displacement relations are expressed by the following equations for large magnitude of deformation.

$$\varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial w}{\partial x}\right)^2\right] \tag{1.3.9}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial y}\right)^2\right] \tag{1.3.10}$$

$$\varepsilon_z = \frac{\partial w}{\partial z} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right] \tag{1.3.11}$$

$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} + \left[\frac{\partial u}{\partial x}\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\frac{\partial v}{\partial y} + \frac{\partial w}{\partial x}\frac{\partial w}{\partial y}\right] \tag{1.3.12}$$

$$\gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} + \left[\frac{\partial u}{\partial y}\frac{\partial u}{\partial z} + \frac{\partial v}{\partial y}\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\frac{\partial w}{\partial z}\right] \tag{1.3.13}$$

$$\gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} + \left[\frac{\partial u}{\partial z}\frac{\partial u}{\partial x} + \frac{\partial v}{\partial z}\frac{\partial v}{\partial x} + \frac{\partial w}{\partial z}\frac{\partial w}{\partial x}\right] \tag{1.3.14}$$

The eqs.(1.3.9 to 1.3.14) are known as Green-Lagrange strain displacement equation. The components of the strain ε_x, ε_y, ε_z, γ_{xy}, γ_{yz} and γ_{zx} define the state of strains in the deformed body, and can be written in a matrix form as

$$\{\varepsilon\}^T = \begin{bmatrix}\varepsilon_x & \varepsilon_y & \varepsilon_z & \gamma_{xy} & \gamma_{yz} & \gamma_{zx}\end{bmatrix} \tag{1.3.15}$$

The relations given in eqs.(1.3.9 to 1.3.14) are non-linear partial differential equations in the unknown component of the displacements. In case of small deformations, the products and squares of the first derivatives are assumed to be negligible compared with the derivatives themselves in many problems of stress analysis. Thus the strain-displacement relations in eqs. (1.3.9 to 1.3.14) reduce to linear relations as follows.

$$
\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} \\
\varepsilon_y &= \frac{\partial v}{\partial y} \\
\varepsilon_z &= \frac{\partial w}{\partial z} \\
\gamma_{xy} &= \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \\
\gamma_{yz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \\
\gamma_{zx} &= \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}
\end{aligned}
\tag{1.3.16}
$$

Eq. (1.3.16) is known as Von-Karman strain displacement equation. The above equation can be expressed in a matrix form as given below.

$$
\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}
\tag{1.3.17}
$$

The above assumption will be incorrect in case of large deformation problems. In these cases, geometric nonlinearity has to be considered.

1.3.3 Linear Constitutive Relations

Hooke's law states that the six component of stress may be described as linear function of six components of strain. The relation for a linear elastic, anisotropic and homogeneous material are expressed as follows.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & . & . & . & C_{16} \\ C_{21} & C_{22} & . & . & . & C_{26} \\ . & . & & & & \\ . & . & & & & \\ . & . & & & & \\ C_{61} & C_{62} & . & . & . & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_{x\varepsilon} \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \tag{1.3.18}$$

or $$\{\sigma\} = [C]\{\varepsilon\} \tag{1.3.19}$$

Where [C] is constitutive matrix. If the material has three orthogonal planes of symmetry, it is said to be orthotropic. In this case only nine constants are required for describing constitutive relations as given below.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{z\tau} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ & C_{22} & C_{23} & 0 & 0 & 0 \\ & & C_{33} & 0 & 0 & 0 \\ & & & C_{44} & 0 & 0 \\ & Symmetry & & & C_{55} & 0 \\ & & & & & C_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \tag{1.3.20}$$

The inverse relation for strains and stresses may be expressed as

$$[\varepsilon] = [C^{-1}]\{\sigma\} = [D]\{\sigma\} \tag{1.3.21}$$

An isotropic is one for which every plane is a plane of symmetry of material behavior and only two constants (Young Modulus, E and Poisson ratio μ) are required to describe the constitutive relation. The following equation includes the effect due to temperature changes as may be necessary in certain cases of stress analysis.

or $$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\mu & -\mu & 0 & 0 & 0 \\ & 1 & -\mu & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & & & 2(1+\mu) & 0 & 0 \\ & & & & 2(1+\mu) & 0 \\ & & & & & 2(1+\mu) \end{bmatrix} \begin{Bmatrix} \sigma_x \\ . \\ . \\ . \\ . \\ \tau_{zx} \end{Bmatrix} + \alpha T \begin{Bmatrix} 1 \\ 1 \\ 1 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \tag{1.3.22}$$

T and α in eq. (1.3.22) denote the difference of temperature and coefficient of thermal expansion respectively.

The inverse relation of stresses in terms of strain components can be expressed as

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \overline{E} \begin{bmatrix} (1-\mu) & \mu & \mu & 0 & 0 & 0 \\ & (1-\mu) & \mu & 0 & 0 & 0 \\ & & (1-\mu) & 0 & 0 & 0 \\ & & & \frac{1-2\mu}{2} & 0 & 0 \\ & & & & \frac{1-2\mu}{2} & 0 \\ & & & & & \frac{1-2\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - \frac{E\alpha T}{1-2\mu} \begin{Bmatrix} 1 \\ 1 \\ 1 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \quad (1.3.23)$$

where $\overline{E} = \frac{E}{(1+\mu)(1-2\mu)}$

1.3.4 Two-Dimensional Stress Distribution

The problems of solid mechanics may be formulated as three-dimensional problems and finite element technique may be used to solve them. In many practical situations, the geometry and loading will be such that the problems may be formulated to two-dimensional or one-dimensional problems without much loss of accuracy. The relation between strain and displacement for two dimensional problems can be simplified from eq. (1.3.16) and can be written as follows.

$$\varepsilon_x = \frac{\partial u}{\partial x}$$
$$\varepsilon_y = \frac{\partial v}{\partial y} \quad (1.3.24)$$
$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

The above expression can be written in a combined form:

$$\frac{\partial^2 \varepsilon_x}{\partial y^2} + \frac{\partial^2 \varepsilon_y}{\partial x^2} = \frac{\partial^2 \gamma_{xy}}{\partial x \partial y} \quad (1.3.25)$$

Eq. (1.3.25) is the compatibility equation since it states the geometric requirements. This condition will ensure adjacent elements to remain free from discontinuities such as gaps and overlaps.

1.3.4.1 Plane stress problem

The plane stress problem is characterized by very small dimensions in one of the normal directions. Some typical examples are shown in Fig. 1.3.5. In these cases, it is assumed that no stress component varies across the thickness and the stress components σ_z, τ_{xz} and τ_{yz} are zero. The state of stress is specified by σ_x, σ_y and τ_{xy} only and is called plane stress.

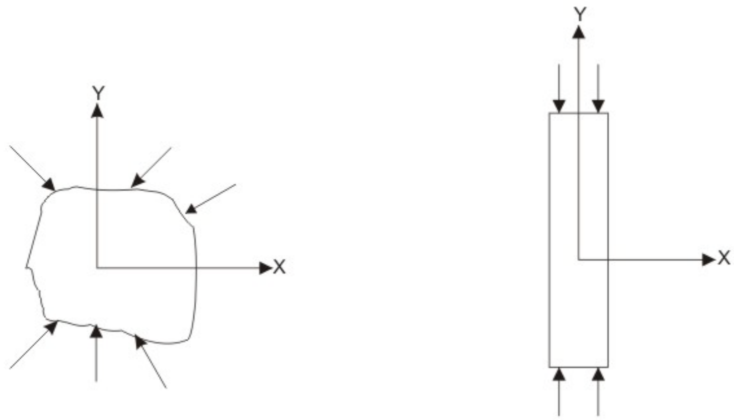


Fig. 1.3.5 Plane stress example: Thin plate with in-plane loading

The stress components may be expressed in terms of strain, which is as follows.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\mu^2} \begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \dfrac{1-\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \frac{E\alpha T}{1-\mu} \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \tag{1.3.26}$$

The strain components can also be expressed in terms of the stress, which is given below.

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\mu & 0 \\ -\mu & 1 & 0 \\ 0 & 0 & 2(1+\mu) \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} + \alpha T \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \tag{1.3.27}$$

It can also be shown that

$$\varepsilon_z = \frac{-\mu}{1-\mu}\left(\varepsilon_x + \varepsilon_y\right) + \frac{1+\mu}{1-\mu}\alpha T \text{ and } \quad \gamma_{yz} = \gamma_{zx} = 0 \tag{1.3.28}$$

1.3.4.2 Plane strain problem

Problems involving a long body whose geometry and loading do not vary significantly in the longitudinal direction are referred to as plane strain problems. Some typical examples are given in Fig. 1.3.6. In these cases, a constant longitudinal displacement corresponding to a rigid body translation and displacements linear in z corresponding to rigid body rotation do not result in strain. As a result, the following relations arise.

$$\varepsilon_z = \gamma_{yz} = \gamma_{zx} = 0 \tag{1.3.29}$$

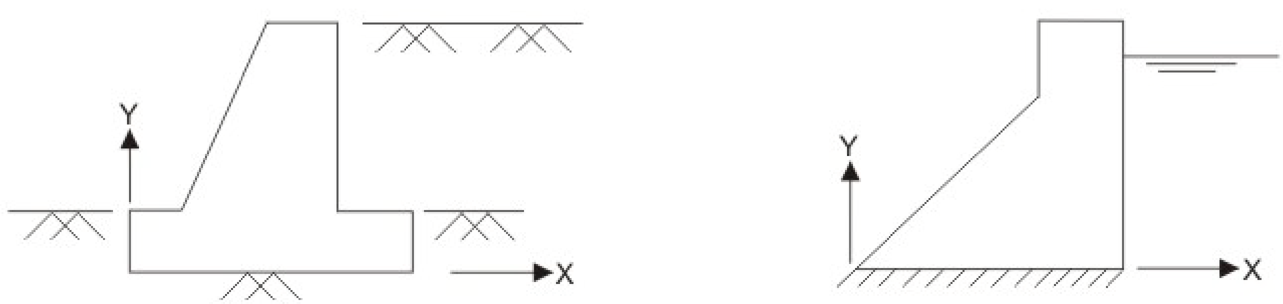
The constitutive relation for elastic isotropic material for this case may be given by,

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\mu)(1-2\mu)} \begin{bmatrix} 1-\mu & \mu & 0 \\ \mu & 1-\mu & 0 \\ 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \frac{E\alpha T}{1-2\mu} \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \tag{1.3.30}$$

Also $\sigma_z = \mu(\sigma_x + \sigma_y) - E\alpha T$ and $\tau_{yz} = \tau_{zx} = 0$ (1.3.31)

The strain components can be expressed in terms of the stress as follows.

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{(1+\mu)}{E} \begin{bmatrix} (1-\mu) & -\mu & 0 \\ -\mu & (1-\mu) & 0 \\ 0 & 0 & 2 \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} + (1+\mu)\alpha T \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \tag{1.3.32}$$



(a) Retaining wall **(b) Dam**

Fig. 1.3.6 Plane strain examples

1.3.4.3Axisymmetric Problem

Many problems in stress analysis which are of practical interest involve solids of revolution subject to axially symmetric loading. A circular cylinder loaded by a uniform internal or external pressure, circular footing resting on soil mass, pressure vessels, rotating wheels, flywheels etc. The strain-displacement relations in these type of problems are given by

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x} \\
\varepsilon_\theta &= \frac{u}{x} \\
\varepsilon_y &= \frac{\partial v}{\partial y} \\
\gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}
\end{aligned} \tag{1.3.33}$$

The two components of displacements in any plane section of the body along its axis of symmetry define completely the state of strain and therefore the state of stress. The constitutive relations are given below for such types of problems.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_\theta \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\mu)(1-2\mu)} \begin{bmatrix} (1-\mu) & \mu & \mu & 0 \\ \mu & 1-\mu & \mu & 0 \\ \mu & \mu & 1-\mu & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_\theta \\ \gamma_{xy} \end{Bmatrix} \tag{1.3.34}$$

Lecture 4: Steps in Finite Element Analysis

1.4.1 Loading Conditions

There are multiple loading conditions which may be applied to a system. The load may be internal and/or external in nature. Internal stresses/forces and strains/deformations are developed due to the action of loads. Most loads are basically "Volume Loads" generated due to mass contained in a volume. Loads may arise from fluid-structure interaction effects such as hydrodynamic pressure of reservoir on dam, waves on offshore structures, wind load on buildings, pressure distribution on aircraft etc. Again, loads may be static, dynamic or quasi-static in nature. All types of static loads can be represented as:

- Point loads
- Line loads
- Area loads
- Volume loads

The loads which are not acting on the nodal points need to be transferred to the nodes properly using finite element techniques.

1.4.2 Support Conditions

In finite element analysis, support conditions need to be taken care in the stiffness matrix of the structure. For fixed support, the displacement and rotation in all the directions will be restrained and accordingly, the global stiffness matrix has to modify. If the support prevents translation only in one direction, it can be modeled as 'roller' or 'link supports'. Such link supports are commonly used in finite element software to represent the actual structural state. Sometimes, the support itself undergoes translation under loadings. Such supports are called as 'elastic support' and are modeled with 'spring'. Such situation arises if the structures are resting on soil. The supports may be represented in finite element modeling as:

- Point support
- Line support
- Area support
- Volume support

1.4.3 Type of Engineering Analysis

Finite element analysis consists of linear and non-linear models. On the basis of the structural system and its loadings, the appropriate type of analysis is chosen. The type of analysis to be carried out depends on the following criteria:

- Type of excitation (loads)
- Type of structure (material and geometry)
- Type of response

Considering above aspects, types of engineering analysis are decided. FEA is capable of using multiple materials within the structure such as:

- Isotropic (i.e., identical throughout)
- Orthotropic (i.e., identical at 90^0)
- General anisotropic (i.e., different throughout)

The Equilibrium Equations for different cases are as follows:

1. Linear-Static:

$$Ku = F \qquad (1.4.1)$$

2. Linear-Dynamic

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = F(t) \qquad (1.4.2)$$

3. **Nonlinear - Static**

$$Ku + F_{NL} = F \qquad (1.4.3)$$

1. Nonlinear-Dynamic

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) + F(t)_{NL} = F(t) \qquad (1.4.4)$$

Here, *M*, *C*, *K*, *F* and *U* are mass, damping, stiffness, force and displacement of the structure respectively. Table 1.4.1 shows various types of analysis which can be performed according to engineering judgment.

Table 1.4.1 Type of analysis

Excitation	**Structure**	**Response**	**Basic analysis type**
Static	Elastic	Linear	Linear-Elastic-Static Analysis
Static	Elastic	Nonlinear	Nonlinear-Elastic-Static Analysis
Static	Inelastic	Linear	Linear-Inelastic-Static Analysis
Static	Inelastic	Nonlinear	Nonlinear-Inelastic-Static Analysis
Dynamic	Elastic	Linear	Linear-Elastic-Dynamic Analysis
Dynamic	Elastic	Nonlinear	Nonlinear-Elastic-Dynamic Analysis
Dynamic	Inelastic	Linear	Linear-Inelastic-Dynamic Analysis
Dynamic	Inelastic	Nonlinear	Nonlinear-Inelastic-Dynamic Analysis

1.4.4 Basic Steps in Finite Element Analysis

The following steps are performed for finite element analysis.

1. **Discretisation of the continuum**: The continuum is divided into a number of elements by imaginary lines or surfaces. The interconnected elements may have different sizes and shapes.
2. **Identification of variables:** The elements are assumed to be connected at their intersecting points referred to as nodal points. At each node, unknown displacements are to be prescribed.
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. The shape or geometry of the element may also be approximated.
4. **Formation of the element stiffness matrix**: After continuum is discretised 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.
5. **Formation of overall stiffness matrix**: 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.
6. **Formation of the element loading matrix:** The loading forms an essential parameter in any structural engineering problem. The loading inside an element is transferred at the nodal points and consistent element matrix is formed.
7. **Formation of the overall loading matrix:** Like the overall stiffness matrix, the element loading matrices are assembled to form the overall loading matrix. This matrix has one column per loading case and it is either a column vector or a rectangular matrix depending on the number of loading cases.
8. **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.
9. **Solution of simultaneous equations:** The unknown nodal displacements are calculated by the multiplication of force vector with the inverse of stiffness matrix.

10. **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. Results may also be obtained by graphical means. It may desirable to plot the contours of the deformed shape of the continuum.

The basic steps for finite element analysis are shown in the form of flow chart below:

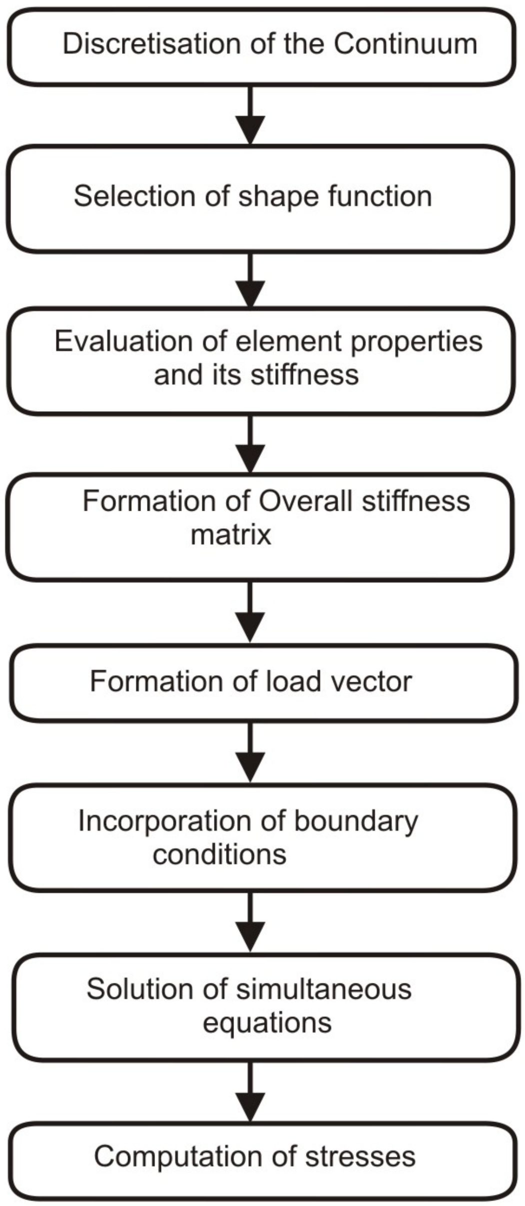


Fig. 1.4.1 Flowchart for steps in FEA

1.4.5 Element Library in FEA Software

A real structure can be modeled with various ways with appropriate assumptions. The structure may be divided into following categories:

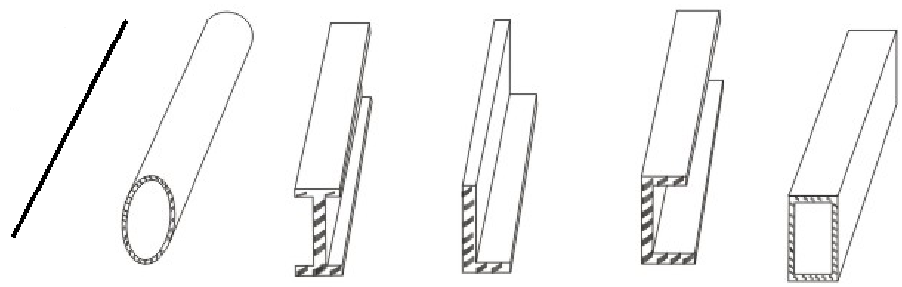
- Cable or tension structures
- Skeletal or framed structures
- Surface or spatial structures
- Solid structures
- Mixed structures

The configuration of structural elements depends upon the geometry of the structural system and the number of independent space coordinates (i.e., x, y and z) required to describe the problem. Thus, the element can be categorized as one, two or three dimensional element. One dimensional element can be represented by a straight line whose ends will be nodal points. The skeletal structures are generally modeled by this type of elements. The pin jointed bar or truss element is the simplest structural element. This element undergoes only axial deformation. The beam element is another type of element which undergoes in-plane transverse displacements and rotations. The frame element is the combination of truss and beam element. Thus, the frame element has axial and in-plane transverse displacements and rotations. This element is generally used to model 1D, 2D and 3D skeletal structural systems. Two-dimensional elements are generally used to model 2D and 3D continuum. These elements are of constant thickness and material properties. The shapes of these elements are triangular or rectangular and it consists of 3 to 9 or even more nodes. These elements are used to solve many problems in solid mechanics such as plane stress, plane strain, plate bending. Three-dimensional element is the most cumbersome which is generally used to model the 3-D continuum. The elements have 6 to 27 numbers of nodes or more. Because of large degrees of freedom, the analysis is time consuming using 3-D elements and difficult to interpret its results. However, for accurate analysis of the irregular continuum, 3-D elements are useful. To analyze any real structure, appropriate elements are to be assigned for the finite element analysis. In standard FEA software, following types of element library are used to discretize the domain.

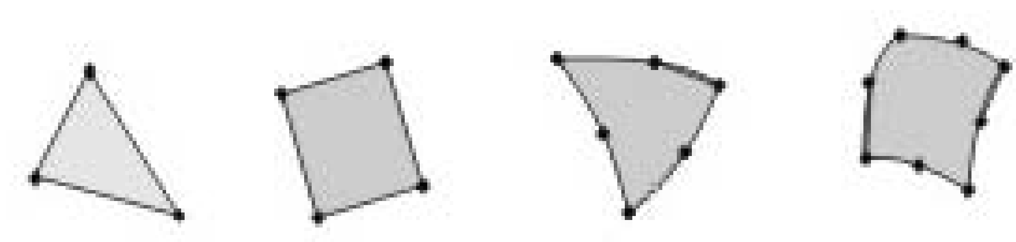
- Truss element
- Beam element
- Frame element
- Membrane/ Plate/Shell element
- Solid element
- Composite element
- Shear panel
- Spring element

- Rigid/Link element
- Viscous damping element

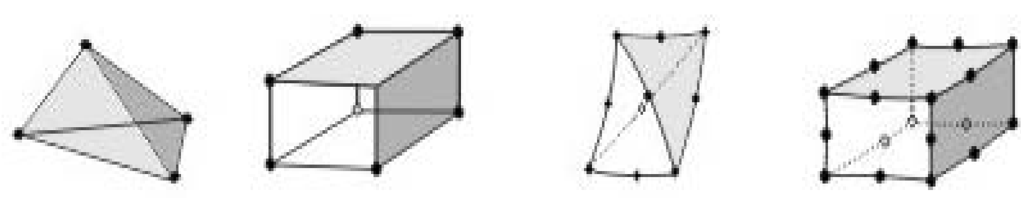
The different types of elements available in standard finite element software are shown in Fig. 1.4.2.



1D Elements (Truss, beam, grid and frame)



2D Elements (Plane stress, Plane strain, Axisymmetric, Plate and Shell)



3D Elements

Fig. 1.4.2 Various types of elements for computer modeling