TERMINOLOGY **UNIT- I**

Finite Element Method (FEM)	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.
Finite Elements	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.
Field Conditions	The field variables, displacements (strains) & stresses or stress resultants must satisfy the governing condition which can be mathematically expressed.
Functional Approximation	A set of independent functions satisfying the boundary conditions is chosen and a linear combination of a finite number of them is taken to approximately specify the field variable at any point.
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.
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	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.
Numerical approach	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.
Finite Difference Method	The application of finite difference method for engineering problems involves replacing the governing differential equations and the boundary condition by suitable algebraic equations.

Nodes	The elements are connected through number of joints which are called Nodes.
Discretization	Discretization is the process of transferring continuous functions, models, variables, and equations into discrete counterparts. This process is usually carried out as a first step toward making them suitable for numerical evaluation.
Continuum	The FEM has been developed to solve problems of continuum mechanics. The continuum will subdivided into finite elements (continuum elements), which can be described with a finite number of parameters. Approach functions are be defined within these elements.
Strain-Displacement	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.
Discretization 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.
Element stiffness matrix	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.

Concepts

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.

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
- Mesh less 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:

$d^4w/dx^4 = q/EI$

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

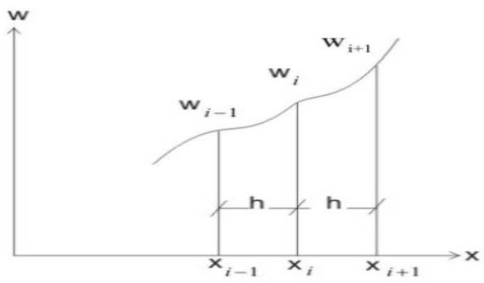


Fig. 1.1.1 Displacement Function

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

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

Thus,

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

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

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

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

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:

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

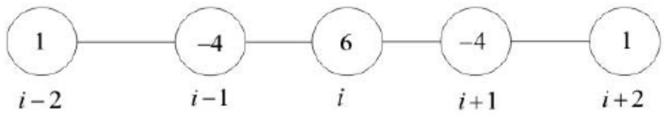


Fig. 1.1.2 Finite difference equation at node i

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.

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.

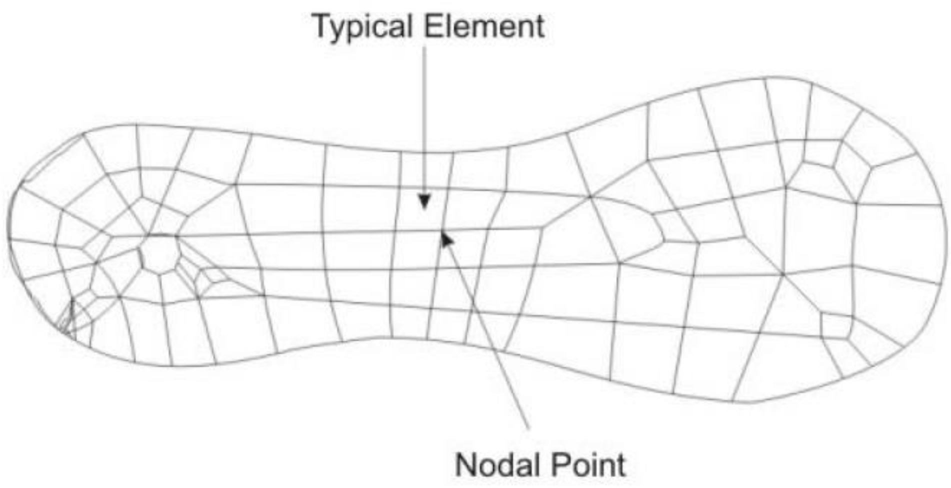


Fig. 1.1.3 Finite element discretization of a domain

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.

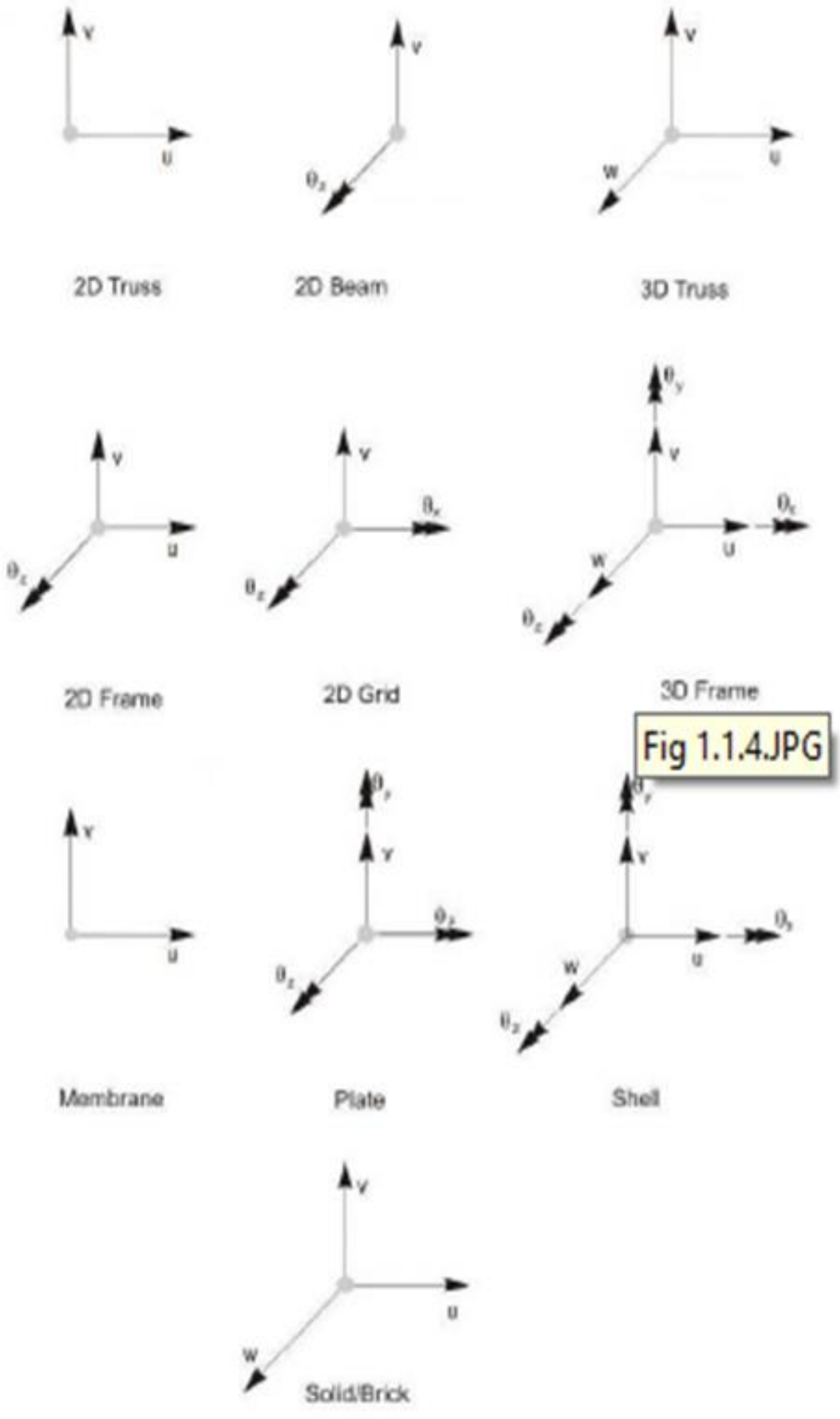
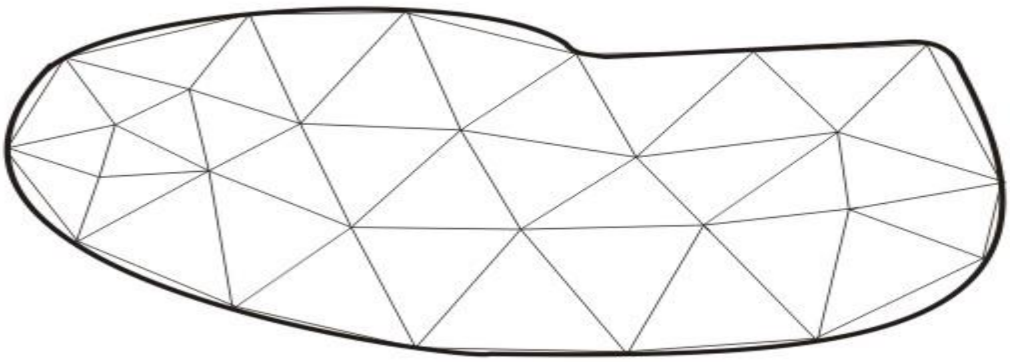


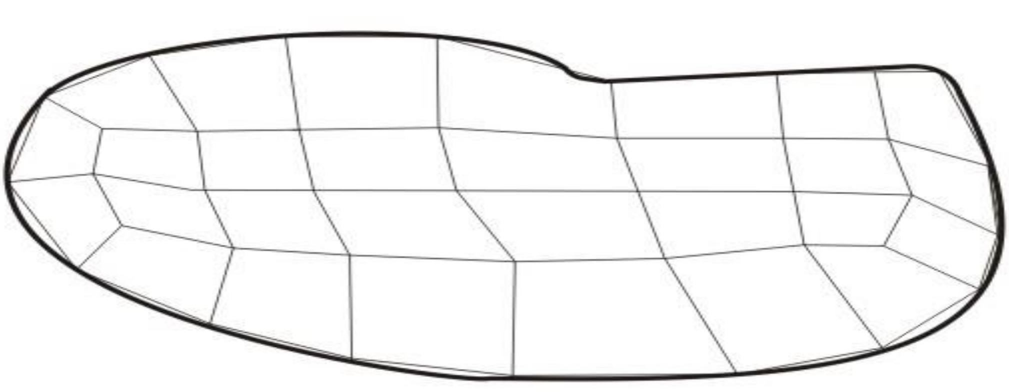
Fig. 1.1.4 Degrees of Freedom for Various Elements

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.



(a) Triangular mesh



(b) Quadrilateral mesh

Fig. 1.2.2 Discretization of a continuum

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.

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.

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.

Important Questions:

1. What are the Steps involved in FEM?
2. Write Merits, Demerits & Limitations of FEM.
3. Describe about Energy Principles.
4. Explain about Discretization Technique.
5. Explain about Rayleigh –Ritz method.
6. Derive Rayleigh –Ritz method of functional approximation.
7. State Equilibrium equations & Derive strain displacement relationships in matrix form.
8. Write Constitutive relationships for

a. Plane stress & Plane strain,

b. And Axi-symmetric bodies of revolution with Axi-symmetric loading.