Module 3 : Method of Analyses

Lecture 15 : Finite Element Method [Section 15.1 : Introduction]

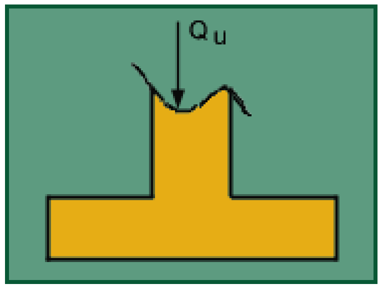
Objectives

In this section you will learn the following

- Introduction

INTRODUCTION

The finite element method is the representation of a body or a structure by an assemblage of sub-divisions called as finite elements. These elements are interconnected at joints which are called nodes or nodal points. Simple functions are chosen to approximate the distribution of actual displacements over each finite element. Such assumed functions are called displacement functions or displacement models. The unknown magnitude of the displacement functions are the displacements at the nodal points. Hence the final solutions will yield the approximate displacements at discrete locations in the body at the nodal points. A displacement model can be expressed in various forms, such as polynomials and trigonometric functions.



Recap

In this section you have learnt the following

- Introduction

Module 3 : Method of Analyses

Lecture 15 : Finite Element Method [Section 15.2: Steps of Analysis]

Objectives

In this section you will learn the following

- Discritization into the element
- Selection of the displacement models
- Derivation of the element stiffens matrix.
- Assembling
- Solution for the unknown displacement
- Computations of the element stresses and strains at the nodal points

The different steps involved in the analysis by Finite Element Method

1. Discritization into the element

The given footing is divided into an equivalent system of finite elements, by a process known as Discritization. The equivalent system may consist of triangular or quadrilateral and/or tetrahedron or hexahedron based on whether the problem is solved as in 2-D or 3-D plane.

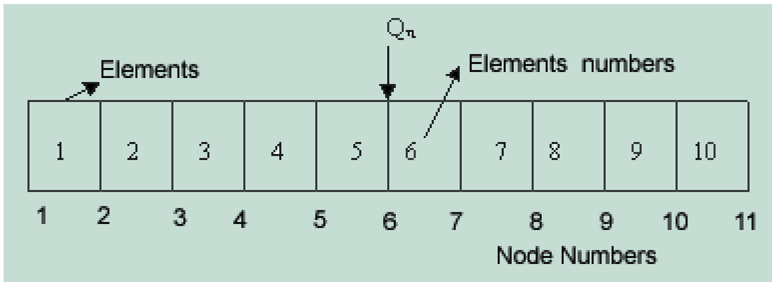


Fig.3.14 Discritization into Elements

2. Selection of the displacement models

It is generally not possible to select a displacement function that can represent exactly the actual variation of displacement in the element. Hence, the basic approximation of the finite element method is introduced at this stage. There are three interrelated factors, which influence the selection of a displacement model.

- The type and degree of displacement model must be chosen.
- The particular displacement magnitude that describes the model must be selected. These are usually the displacement at nodal points, but they may also include the derivation of the displacement at some or all of the nodal points.
- The displacement model must specify certain requirements, which ensure that the numerical result approach the correct solution.

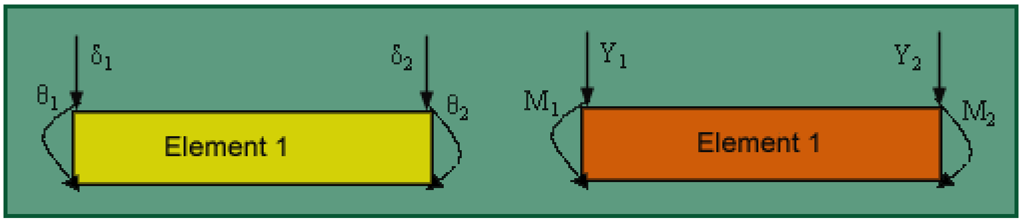


Fig.3.15 Displacements and forces acting at corresponding nodes

$$\text{Displacement matrix, } \delta = \begin{Bmatrix} \delta_1 \\ \theta_1 \\ \delta_2 \\ \theta_2 \end{Bmatrix} \text{Force Matrix, } F = \begin{bmatrix} Y_1 \\ M_1 \\ Y_2 \\ M_2 \end{bmatrix}$$

3. Derivation of the element stiffens matrix.

The stiffness matrix consists of the co-efficient of the equilibrium equations derived from the material and properties of an element and obtained by the use of minimum potential energy. The stiffness relates the displacements at the nodal points (the nodal forces) to the applied forces at the nodal points. The distributed forces applied to the structure are converted into equivalent concentrated forces at nodes. The equilibrium relates the stiffness matrix [K], nodal force vector [F], and the nodal displacement vector [d] is expressed as a set of simultaneous linear algebraic equations.

Element stiffness matrix, $K_{11} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$

4. Assembling

This process includes the assemblage of the overall or global stiffness matrix for the entire body from the individual element stiffness matrices and the overall or global forces or load victor from the element nodal vectors. The most common assemblage technique used is called as the direct stiffness matrix [K].

[K] {d}=[F] Global stiffness matrix, $K_g = \begin{bmatrix} K_1 & & & \\ & K_2 & & \\ & & \dots & \\ & & & K_n \end{bmatrix}$

5. Solution for the unknown displacement

The algebraic equations obtained in the above step are solved for the unknown displacements. In linear equilibrium equations, this is a straightforward application of matrix algebra techniques. However, for nonlinear problems the desired solutions are obtained by sequence of steps, each step involving the modification of the stiffness matrix and/ or load vector.

Therefore displacements at nodal points are, $\delta = K_g^{-1} F$

6. Computations of the element stresses and strains at the nodal points

In some cases the magnitude of the primary unknowns, that is the nodal displacements, will be all that is required for an engineering solution. More often, however, the other quantities derived from the primary unknown like stresses, strains must be computed.

Force acting on each element is, $F_1 = K_1 \delta$

Recap

In this section you have learnt the following

- Discritization into the element
- Selection of the displacement models
- Derivation of the element stiffens matrix.
- Assembling
- Solution for the unknown displacement
- Computations of the element stresses and strains at the nodal points

Congratulations, you have finished Lecture 15. To view the next lecture select it from the left hand side menu of the page

Recap

In this section you have learnt the following

- Discritization into the element
- Selection of the displacement models
- Derivation of the element stiffens matrix.
- Assembling
- Solution for the unknown displacement
- Computations of the element stresses and strains at the nodal points

Congratulations, you have finished Lecture 15. To view the next lecture select it from the left hand side menu of the page