Universiti Malaysia PAHANG
Engineering • Technology • Creativity

Finite Element Analysis

Introduction

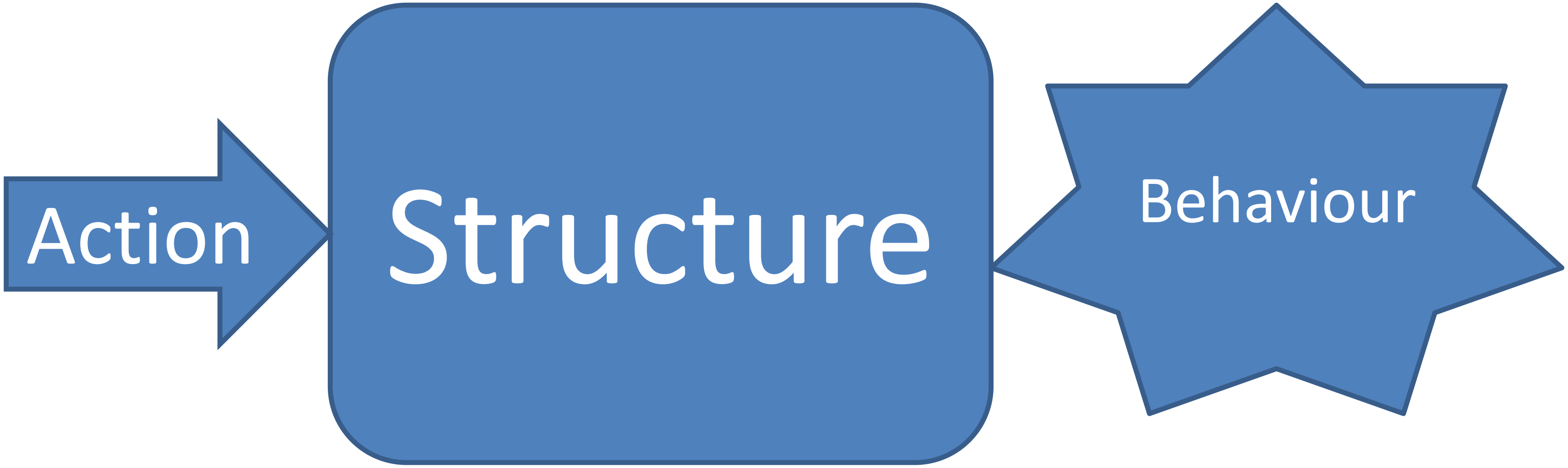
by
Dr. Gul Ahmed Jokhio
Faculty of Civil Engineering and Earth Resources
jokhio@ump.edu.my

Lesson Outcomes

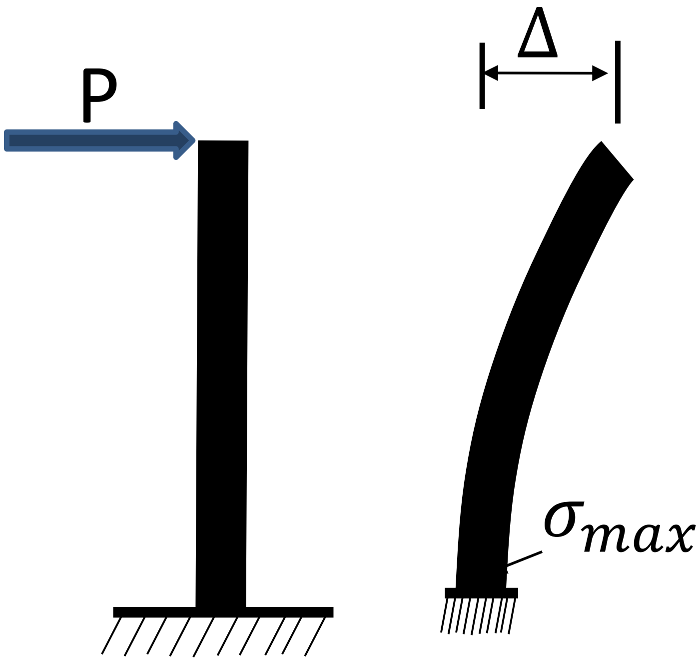
- At the end of this lecture, the student should be able to:
  - Define the Finite Element Method
  - Define Discretization process
  - Understand applications of Finite Element Method
- Name general Finite Element Approaches
- Describe the 8 steps of Finite Element Analysis

Structural Analysis

The primary objective of analyzing a structure is to find out how it will behave when subjected to a given action.



Mathematical Modelling of Structures



Known Quantities

Length, L; Modulus of Elasticity, E; Strength, fy; Width, b; Depth, d.

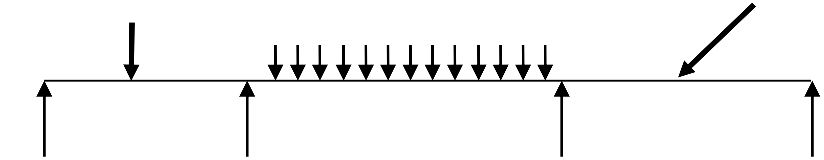
Unknown Quantities

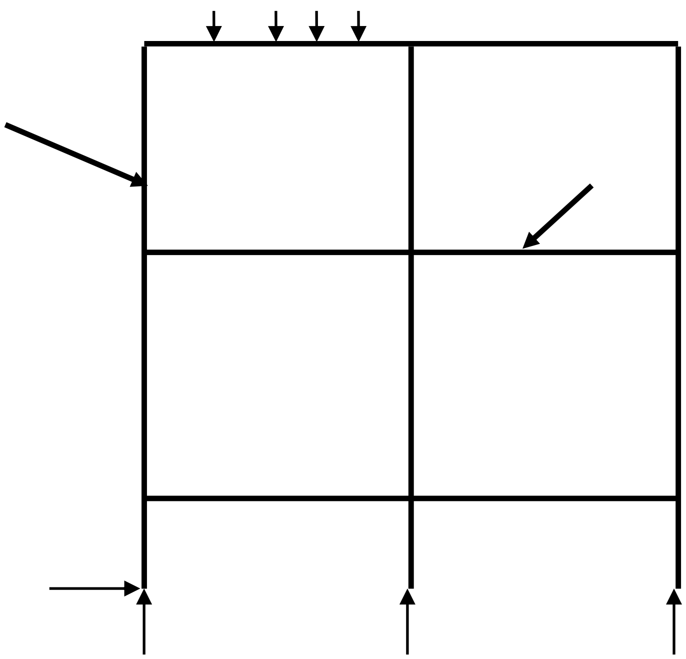
$$\Delta = \frac{PL^3}{3EI}$$

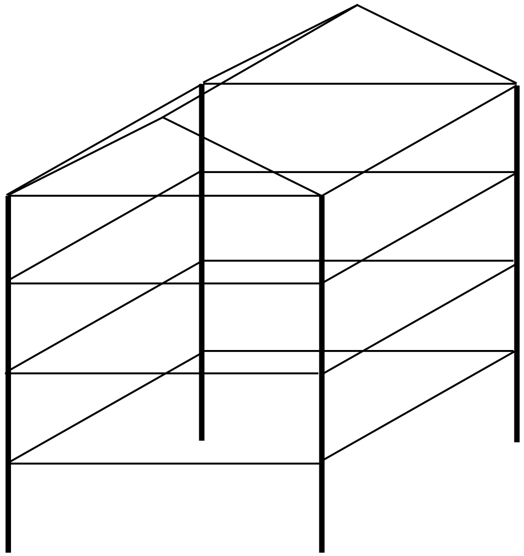
$$\sigma_{max} = \frac{PL}{S}$$

A physical structure can be converted into a mathematical model in the form of equations for the known quantities

Increasing Complexity of Structural Analysis







Simple structures can be analyzed easily, but as the structure complexity increases, the number and the complexity of mathematical equations for the solution will also increase making it impractical.

Complexities in Engineering Problems

- Geometry
  - Initial geometry
  - Change in geometry
- Material Properties
  - Initial material properties
  - Changing material properties
- Loading
  - Initial loading conditions
  - Changing loading conditions
  - Vibrations, Impacts, etc.

Apart from the size of the structure, the above complexities add to the difficulty in analyzing structures.

Finite Element Method

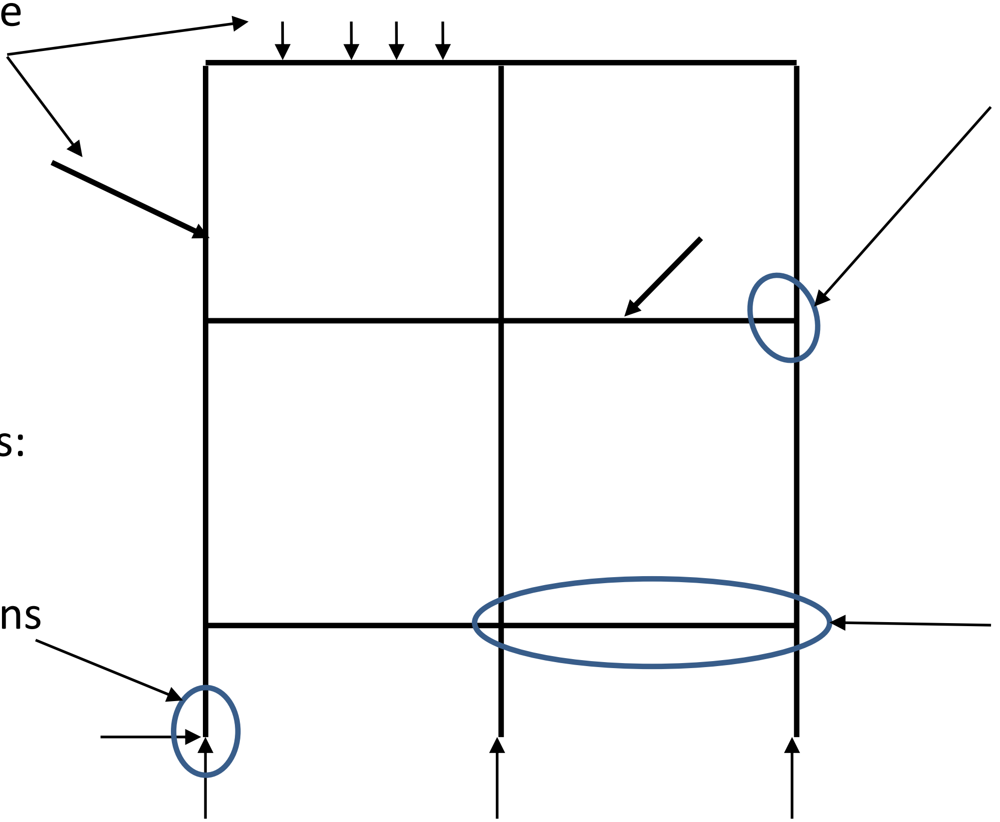
For large structures involving many complexities, the Finite Element Method is a good alternative to analytical methods.

- Numerical Method (As opposed to Analytical Method)
- Approximate Solution
- Solution at Discrete Points only
- Discretizes the structure into nodes, elements, and boundary conditions

Discretization

Loads acting on the structure

Nodes: Interconnecting points common to two or more elements



Boundary Conditions: Restrictions on movement (translations/rotations

Elements: Equivalent smaller bodies or units

Applications of Finite Element Method

- Structural Engineering (Stress Analysis)
  - Static Analysis
  - Dynamic Analysis
- Heat-transfer
- Fluid Mechanics
- Electrostatics

Stress Analysis Problem

- Given
  - Structure
    - Geometry
    - Material Properties
    - Boundary Conditions
  - Applied Loading
- Sought
  - Displacements
  - Stresses

General FEM Approaches

- Force or Flexibility Method
  - Internal forces taken as unknowns
  - First equilibrium equations are used
  - Necessary additional equations found by introducing compatibility
  - Results in a set of algebraic equations
  - $\Delta = KF$
  - Here, K is the flexibility matrix

General FEM Approaches

- Displacement or Stiffness Method
  - Nodal displacements are taken as unknowns
  - Compatibility conditions are imposed
  - Governing equations are obtained from equilibrium
  - Results in a set of algebraic equations
  - $F = K\Delta$
  - Here, K is the stiffness matrix

Simpler formulation, more desirable
Widely used in general purpose FEA

Steps in Finite Element Analysis

1. Discretize and select element types
2. Select a displacement function
3. Define the strain/displacement and stress/strain relationships
4. Derive the element stiffness matrix and equations
5. Assemble the element equations to obtain the global or total equations and introduce boundary conditions
6. Solve for the unknown degrees of freedom or generalized displacements
7. Solve for the element strains and stresses
8. Interpret the results

Step 1(a): Discretization

- Dividing the body into equivalent system of finite elements
- Choosing element types that most closely model the actual physical behaviour
- Choose appropriate number of elements
  - Larger the elements, the more in-accurate are the results
  - Smaller elements; therefore, a larger number of these increases computational effort

Step 1(b): Element Types

- Choose element type suitable for the structure
- Line Elements (One Dimensional)
  - Typically used to model prismatic structural members such as beams and columns.
  - Can have 2 or 3 nodes
- Area Elements (Two Dimensional)
  - Typically used to model area type structural members such as slabs, walls, etc.
  - Simplest triangular elements have 3 nodes whereas, quadrangular elements can have up to 8 nodes

Step 1(b): Element Types (Continued)

- Volume Elements (Three Dimensional)
  - These are used to model any kind of structural member when more detailed results are required.
  - Generally known as ‘brick elements’, these can have from 4 and up to 20 nodes
  - These provide more accuracy but are typically very expensive in terms of computational cost

Step 2: Displacement Functions

- A function defining the displacement within the element using nodal values
- Also called a shape function
- Approximates the displacement at any point within the element as a function of the nodal values
- Polynomial
  - Linear
    - For example $u = a_1 + a_2 x$
  - Quadratic
    - For example $u = a_1 + a_2 x + a_3 x^2$
  - Cubic etc.
- Trigonometric series, etc.

Step 3(a): Strain-Displacement Relationship

- A mathematical equation relating strain at any point within the element to the displacement at that point
- For example, for a one dimensional deformation, for small strains
- $\varepsilon_x = \dfrac{du}{dx}$

Step 3(b): Stress-Strain Relationship

- Generally called ‘Constitutive Law’
- Stress as a mathematical function of strain
- Hooke’s law is the simplest form of stress-strain relationship
- For example, for a linear elastic material
- $\sigma_x = E\varepsilon_x$

Step 4(a): Element Stiffness Matrix

- Direct Equilibrium or Stiffness Method
  - Relates nodal forces to nodal displacements
  - Suitable for line elements
- Work or Energy Methods
  - Uses principle of virtual work, minimum potential energy etc.
  - Suitable for area and volume elements
- Methods of Weighted Residuals
  - Popular in Galerkin's method
  - Useful when a functional such as potential energy is not readily available

Step 4(b): Element Equations

- Element equations in typical form are represented as:
- $$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ \vdots \\ f_n \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & \cdots & k_{1n} \\ k_{21} & k_{22} & k_{23} & \cdots & k_{2n} \\ k_{31} & k_{32} & k_{33} & \cdots & k_{3n} \\ \vdots & & & & \vdots \\ k_{n1} & \cdots & \cdots & \cdots & k_{nn} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ \vdots \\ d_n \end{Bmatrix}$$
- Compact matrix form:
- $\{f\} = [k]\{d\}$
- $\{f\}$ is the vector of element nodal forces
- $[k]$ is the element stiffness matrix
- $\{d\}$ is the vector of unknown element nodal degrees of freedom or generalized displacements (May include actual displacements, slopes, or curvatures)

Step 5(a): Assembly of Element Equations

- $\{F\} = [K]\{d\}$
- Where,
- $\{F\}$ is global nodal forces vector, should be same as applied loads for equilibrium
- $[K]$ is structure global or total stiffness matrix
- $\{d\}$ is the vector of known or un-known structure nodal degrees of freedom
- For most problems, $[K]$ is square and symmetric
- $[K]$ is also singular as its determinant is zero
- To remove the singularity, boundary conditions are applied

Steps 5(a): Assembly of Element Equations (Continued)

- The structure stiffness matrix is obtained as:
- $[K] = \sum_{e=1}^{N} \left[k^{(e)}\right]$
- And the structure force matrix is obtained as:
- $\{F\} = \sum_{e=1}^{N} \left\{f^{(e)}\right\}$
- $\sum$ here means assembly as per the element connectivity
- $\{F\} = [K]\{d\}$
- Boundary conditions are then applied and the resulting system of equations is solved
- Element forces are then found by back-substitution

Step 5(b): Boundary Conditions

- Suppose the system of equations for a structure has been assembled as:
- $\begin{Bmatrix} F_{1x} \\ F_{2x} \\ F_{3x} \end{Bmatrix} = \begin{bmatrix} k_1 & 0 & -k_1 \\ 0 & k_2 & -k_2 \\ k_1 & -k_2 & k_1 + k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$
- $[K]$ in the above equation is singular i.e. its determinant is zero. This system of equations can only be solved after applying boundary conditions

Step 5(b): Boundary Conditions (Continued)

- There are two types of boundary conditions
  - Primary, essential, or Dirichlet (Johann Dirichlet, 1805-1859)
    - Specifies displacement values for certain degrees of freedom
    - Called ‘homogenous’ when these specified values are zero, otherwise called ‘non-homogeneous’ boundary conditions
  - Natural or Neumann (Carl Neumann, 1832-1925)
    - Specifies the values that the derivatives of a solution must satisfy

Step 6: Solution of System of Equations

- Once the boundary conditions have been applied, the system of equations can be solved
- The final system of equations obtained in the Finite Element Analysis is a linear system of simultaneous equations
- There are many methods available for the solution of a such a system, such as the Frontal Method, the PCD Solver, etc.
- Most of these methods are aimed at minimizing the computational effort depending upon the particular shape of the structural stiffness matrix

Step 6: Solution of System of Equations (Continued)

- Therefore, different solution methods perform best for only particular types of problems
- Most of these solution methods are based on Gaussian elimination
- In this course we will use the Gaussian elimination in its simplest form i.e. performing row operations on the augmented matrix to bring it into upper triangular form
- An augmented matrix is obtained by tagging the left hand side ‘force’ vector to the right as an additional column to the stiffness matrix

Step 7: Element Strains and Stresses

- Once the nodal displacements for the structure are known, these can be applied into the element equations to find element forces
- From the forces, the determination of element stresses is straight forward
- Once element stresses are known, the stress-strain relationships can be used to determine the element strains
- Also, the strain-displacement relationships can be used to determine the displacement within an element

Step 8: Interpretation of Results

- The Finite Element Analysis provides results in terms of nodal displacements only
- From these nodal displacements, element stresses and strains can be determined, which are known as the ‘derived results’
- How these results are interpreted is completely on the discretion of the engineer
- Different codes provide guidelines and limitations for a structure to be considered ‘safe’
- These limitations are generally imposed upon the values of maximum stresses or ‘deflections’
- The engineer can use their judgement to decide the suitability of the structure

Author Information

Dr. Gul Ahmed Jokhio

is a Senior Lecturer at FKASA, UMP. He completed his PhD from Imperial College London in 2012.