SNS COLLEGE OF TECHNOLOGY

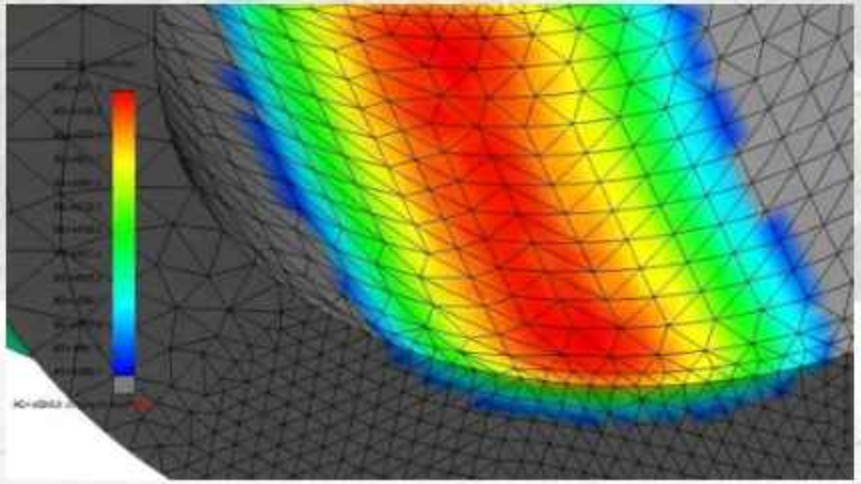
(AN AUTONOMOUS INSTITUTION)

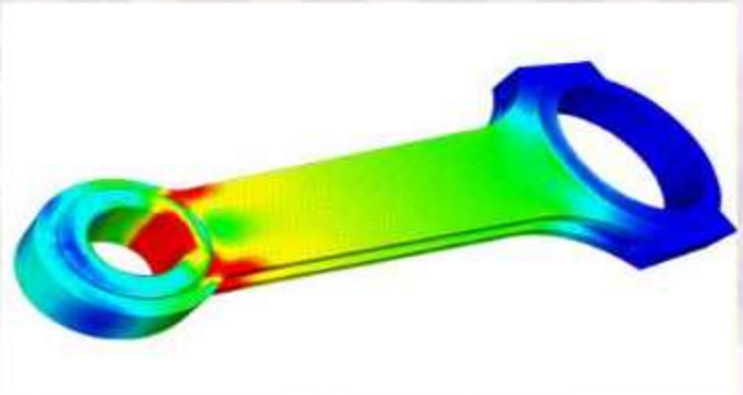
Department of Mechanical Engineering

FINITE ELEMENT ANALYSIS

Unit – I

GENERAL STEPS INVOLVED IN FINITE ELEMENT ANALYSIS





Prepared by

P.Janagarathinam,

Assistant Professor / Mechanical Engineering

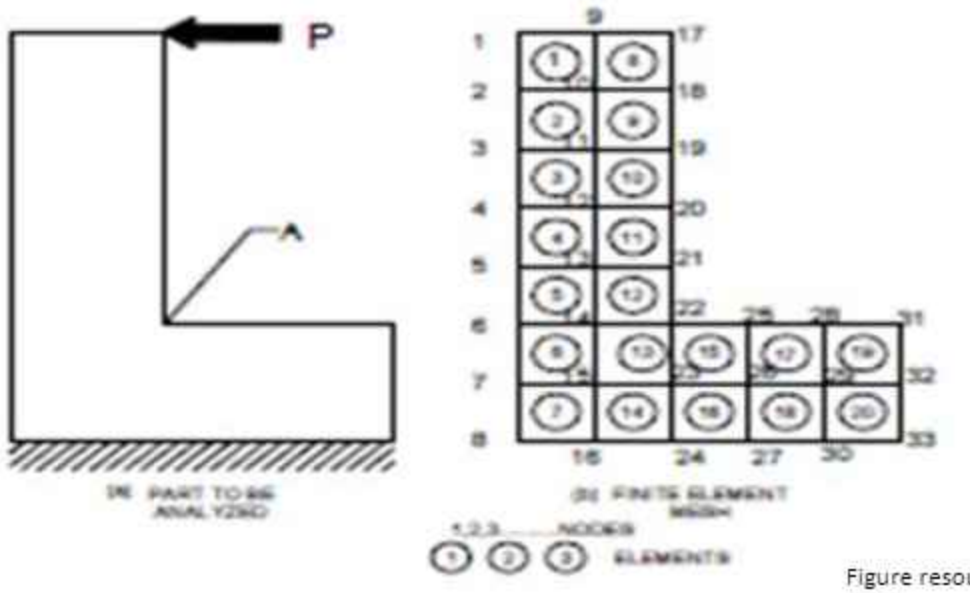
SNS College of Technology, Coimbatore

HISTORY OF FEA

- FEA was initially developed in 1943 by R. Courant to obtain approximate solution to vibration problems
- Turner et al published in 1956 a paper on “Stiffness and Deflection of Complex Structures”

WHAT IS FINITE ELEMNT METHOD ?

- The finite element method is a numerical procedure. This method involves modeling the structure using a finite number of small interconnected elements. Consider the plate shown in Fig.



Finite Element Modeling

Figure resource : Introduction to FEA Book

GENERAL STEPS INVOLVED IN FINITE ELEMENT ANALYSIS

- **Step 1. Select The Element Type And Discretize The Component**

The first step is to select an element which closely represents the physical behavior of the structure..

- **Step 2 . Select A Displacement Function**

A displacement function within the element using the nodal values of the element is then defined. These may be linear, quadratic or cubic polynomials. The same displacement function is used for all the elements.

- **Step 3. Define Stress Strain Relationship**

Definition of stress strain relationship for each element is the next step. For example in the case of one dimensional deformation, if u is the displacement in the direction X, the strain is related to the displacement by the relation

$$\varepsilon_x = du/dx$$

- The stresses are related to the strains through the stress strain law or constitutive law. Using Hooke's law, the stress strain law can be written as:

$$\sigma_x = E\varepsilon_x$$

where σ is the stress in the X direction and E is the modulus of elasticity.

Step 4. Derive The Element Stiffness Matrix

The stiffness matrix could be derived by the direct equilibrium method or work or energy method or method of weighted residuals.

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ - \\ - \\ f_n \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & - & k_{1n} \\ k_{21} & k_{22} & k_{23} & - & - \\ k_{31} & k_{32} & - & - & - \\ - & - & - & - & - \\ - & - & - & - & - \\ k_{n1} & - & - & - & k_{nn} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ - \\ - \\ d_n \end{Bmatrix}$$

This may also be written in the form {f} = [k] {d} where {f} is the vector of element nodal forces, [k] is the element stiffness matrix and {d} is the displacement vector.

- **Step.5. Assemble Global Stiffness Matrix**

The individual element stiffness matrices are then assembled to obtain the global stiffness matrix of the whole component being analyzed.

- **Step 6. Solve To Obtain Nodal Displacements**

Solve the below equation to get displacement values

$$[F] = [K]\{d\}$$

- **Step.7. Solve For Element Strains And Stresses**

Strains can be computed from the displacements. Once the strain is known the stress can be calculated using Hooke's law. Principal stresses, shear stresses, von Mises stresses (equivalent stress) etc. could be computed depending on the interest of the designer.

TYPES OF ANALYSIS

- Structural Analysis (Static And Dynamic Analysis)
- Linear And Non-linear Analysis
- Thermal Analysis
- Fluid Flow Analysis
- Field Analysis (Electrical, Magnetic, Electromagnetic And Electrostatic)

Multible Choice Questions

- The solution by FEM is

  a)always exact **b)mostly approximate**

  c)sometimes exact d)never exact

- Accuracy of solution __ with increase of number of elements

  a) Improves b)reduces c)no change d)depends on other data

- A numerical technique for finding approximate solutions of partial differential equations as well as of integral equations.

  a) Hybrid method b) mixed method **c) finite element method** d) Finite volume method

THANK YOU