Al-Mustaqbal University

College of Engineering & Technology

Biomedical Engineering Department

Subject Name: CAD 2

4th Class, Second Semester

Subject Code: [MU0114205]

Academic Year: 2024-2025

Lecturer: م.د علي كامل كريم

Email: ali.kamil.kareem@uomus.edu.iq

Lecture No.3

Lecture Title: [Finite element analysis software (FEA software)

https://eng.uomus.edu.iq/DefaultDep.aspx?depid=13

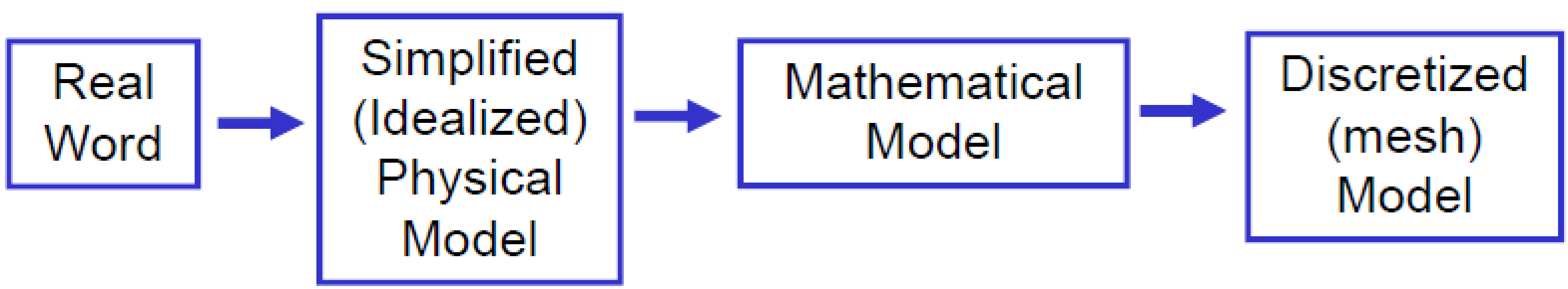
FEA

Discretization

Complex Object **Simple Analysis**

(Material discontinuity,

Complex and arbitrary geometry)



- Continuum mechanics is a branch of mechanics that deals with the deformation of and transmission of forces through materials modeled as a continuous mass rather than as discrete particles such as solid mechanic, fluid , thermodynamic … etc.

- 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.

Discretizations

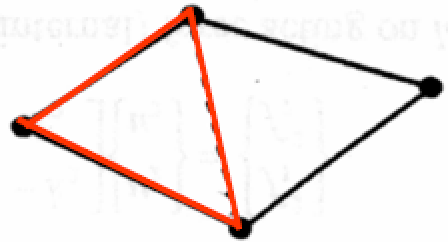
- Model body by dividing it into an equivalent system of many smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces.

Types of Finite Elements

1-D (Line) Element

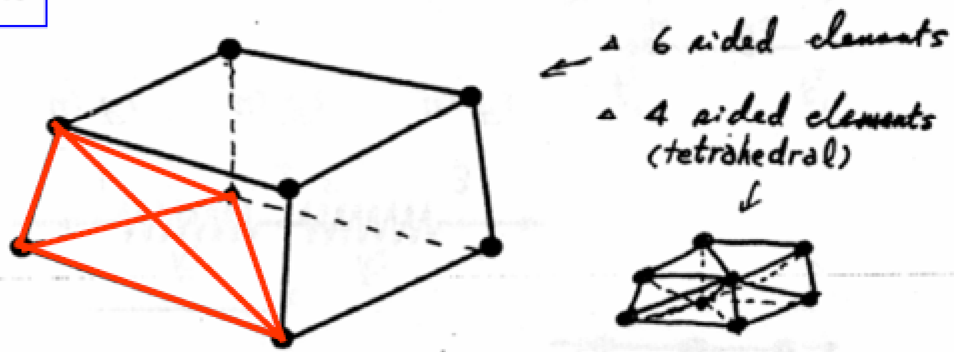
(Spring, truss, beam, pipe, etc.)

2-D (Plane) Element



(Membrane, plate, shell, etc.)

3-D (Solid) Element



(3-D fields - temperature, displacement, stress, flow velocity)

Elements & Nodes - Nodal Quantity

Shape Functions

The values of the field variable computed at the nodes are used to approximate the values at non-nodal points (that is, in the element interior) by *interpolation* of the nodal values. For the three-node triangle example, the field variable is described by the approximate relation

$$\varphi(x, y) = N_1(x, y)\, \varphi_1 + N_2(x, y)\, \varphi_2 + N_3(x, y)\, \varphi_3$$

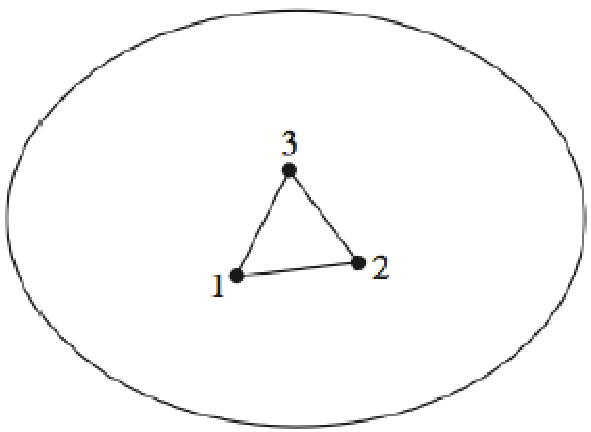
where φ_1, φ_2, and φ_3 are the values of the field variable at the nodes, and N_1, N_2, and N_3 are the ***interpolation functions***, also known as ***shape functions*** **or** ***blending functions***.

In the finite element approach, the nodal values of the field variable are treated as unknown *constants* that are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes.

The interpolation functions are predetermined, *known* functions of the independent variables; and these functions describe the variation of the field variable within the finite element.

Degrees of Freedom

Again a two-dimensional case with a single field variable $\varphi(x, y)$. The triangular element described is said to have *3 degrees of freedom,* as three nodal values of the field variable are required to describe the field variable everywhere in the element (scalar).



$$\varphi(x, y) = N_1(x, y)\,\varphi_1 + N_2(x, y)\,\varphi_2 + N_3(x, y)\,\varphi_3$$

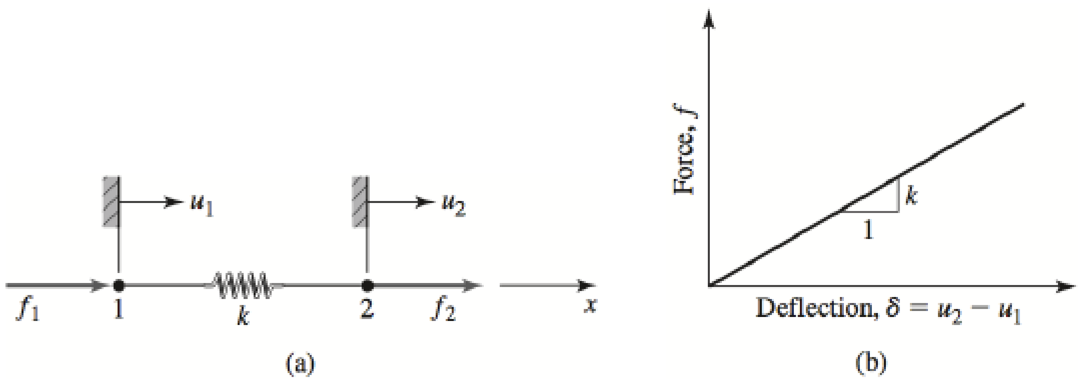
In general, the number of *degrees of freedom* associated with a finite element *is equal to the product of the number of nodes and the number of values of the field variable* (and possibly its derivatives) that must be computed at each node.

Stiffness Matrix

The primary characteristics of a finite element are embodied in the element ***stiffness matrix***. For a structural finite element, the stiffness matrix contains the geometric and material behavior information that indicates the resistance of the element to deformation when subjected to loading. Such deformation may include axial, bending, shear, and torsional effects. For finite elements used in nonstructural analyses, such as fluid flow and heat transfer, the term *stiffness matrix* is also used, since ***the matrix represents the resistance of the element to change when subjected to external influences***.

LINEAR SPRING AS A FINITE ELEMENT

A linear elastic spring is a mechanical device capable of supporting axial loading only, and the elongation or contraction of the spring is directly proportional to the applied axial load. The constant of proportionality between deformation and load is referred to as the *spring constant, spring rate,* or ***spring stiffness k***, and has units of force per unit length. As an elastic spring supports axial loading only, we select an *element coordinate system* (also known as a *local* coordinate system) as an x axis oriented along the length of the spring, as shown.



(a) Linear spring element with nodes, nodal displacements, and nodal forces.
(b) Load-deflection curve.

Assuming that both the nodal displacements are zero when the spring is undeformed, the net spring deformation is given by

$\delta = u_2 - u_1$

and the resultant axial force in the spring is

$f = k\delta = k(u_2 - u_1)$

For equilibrium,

$f_1 + f_2 = 0$ or $f_1 = -f_2$,

Then, in terms of the applied nodal forces as

$\boldsymbol{f_1 = -k(u_2 - u_1)}$

$\boldsymbol{f_2 = k(u_2 - u_1)}$

which can be expressed in matrix form as

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \quad \text{or} \quad [k_e]\{u\} = \{f\}$$

where

$$[k_e] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

Stiffness matrix for one spring element

is defined as the element stiffness matrix in the element coordinate system (or local system), $\{u\}$ is the column matrix (vector) of nodal displacements, and $\{f\}$ is the column matrix (vector) of element nodal forces.

$$\begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} = [k_e] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \qquad \text{with} \qquad [k_e] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

known $\{F\} = [K]\ \{X\}$ unknown

The equation shows that the element stiffness matrix for the linear spring element is a ***2 × 2 matrix***. This corresponds to the fact that the element exhibits ***two nodal displacements (or degrees of freedom)*** and that ***the two displacements are not independent (that is, the body is continuous and elastic)***.

FEA for multiple (many) elements

$$\{F\} = [K] \cdot \{U\}$$

- $\{F\}$: Array of applied forces (one for each DOF)
- $[K]$: Matrix of stiffnesses (DOF x DOF)
- $\{U\}$: Array of displacements (one for each DOF)

$\{F\}$ is "known" (loads)

$[K]$ is "known" (geometry, material properties…elements)

$\{U\}$ is to be determined (displacements)

This can be solved mathematically using a matrix inversion method

$$\{F\} = [K] \cdot \{U\} \quad \rightarrow \quad \{U\} = [K]^{-1}\{F\}$$

(first nodal quantity)

Once the displacements $\{U\}$ are known, then strains and stresses can be determined:

$$\varepsilon = \frac{\Delta u}{L} \text{ (1-D …more complicated for 2-D and 3-D strains)}$$

$$\sigma = E \cdot \varepsilon$$

and $FOS = \frac{\sigma_y}{\sigma}$

(second nodal quantities)