Undergraduate Course

# FINITE ELEMENT METHODS

## Introduction

Dr. Bashar Salhab

جَـامعة المَـنارة
MANARA UNIVERSITY

1

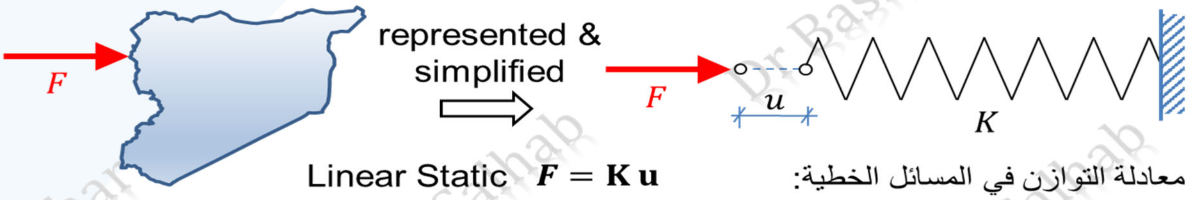


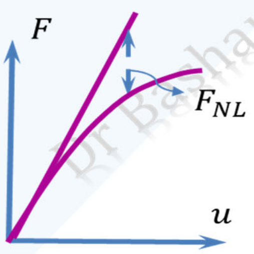
## Structural Analysis

Methods include: Force method (ex. Virtual work), Displacement methods (ex. Slope & deflection), Finite element method (ex. Robot)

- Structural Analysis
  - Linear elastic Analysis
  - Nonlinear Analysis
    - Geometrically
    - Materially

- Structural Analysis
  - Static Analysis
  - Dynamic Analysis




Linear Static $\boldsymbol{F} = \mathbf{K}\,\mathbf{u}$ معادلة التوازن في المسائل الخطية:

Nonlinear Static $\boldsymbol{F} = \mathbf{K}\,\mathbf{u} - \boldsymbol{F}_{NL}$ معادلة التوازن في المسائل غير الخطية:

Linear Dynamic $\boldsymbol{F}(t) = \mathbf{m}\,\ddot{\mathbf{u}}(t) + \mathbf{c}\,\dot{\mathbf{u}}(t) + \mathbf{K}\,\mathbf{u}(t)$

معادلة التوازن في المسائل الديناميكية غير الخطية (المعادلة العامة):

Nonlinear Dynamic $\boldsymbol{F}(t) = \mathbf{m}\,\ddot{\mathbf{u}}(t) + \mathbf{c}\,\dot{\mathbf{u}}(t) + \mathbf{K}\,\mathbf{u}(t) - \boldsymbol{F}_{NL}(t)$

2

External effect from nature estimated by norms

⇓

**Excitation:**

Loads, Vibrations, Settlements, Thermal changes

⇒

**Structure**

(structural analysis model)
Element = material + dimensions

Types: elastic or plastic

⇒

**Responses:**

Internal effect: Displacements
$\{\mathbf{u}\} = [\mathbf{K}]^{-1}\{\mathbf{F}\}$
$\longrightarrow$ Stress, Strain

⇑

Structural design/check

Types: Static or dynamic

Types: linear or nonlinear

***For example, Linear Static Analysis:***

Linear analysis can provide most of the information about the behavior of a structure, and can be a good approximation for many analyses. It is also the bases of nonlinear analysis in most of the cases.

Most structural analysis problems can be treated as linear static problems, based on the following assumptions:

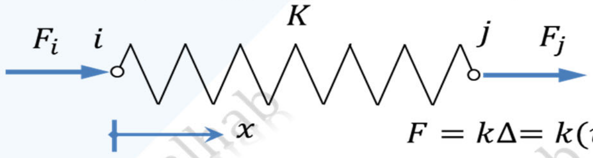
1. Small deformations (loading pattern is not changed due to the deformed shape)
2. Elastic materials (no plasticity or failures)
3. Static loads (the load is applied to the structure in a slow or steady state)

3

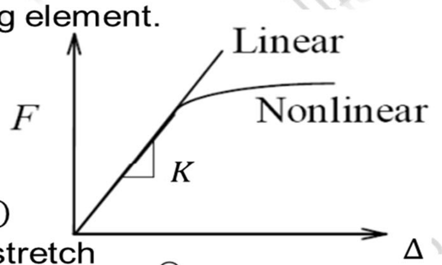
***Matrix displacement methods of structural analysis***

The method is based on the elastic theory, where it can be assumed that most structures behave like complex elastic springs, the load-displacement relationship of which is linear. Obviously, the analysis of such complex springs is extremely difficult, but if the complex spring is subdivided into a number of simpler springs, which can readily be analyzed, then by considering equilibrium and compatibility at nodes, the entire structure can be represented by a large number of simultaneous equations. Solution of the simultaneous equations results in the displacements at these nodes, then stresses can be obtained in each individual spring element.

Everything important is simple

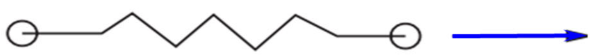



$F = k\Delta = k(u_j - u_i)$



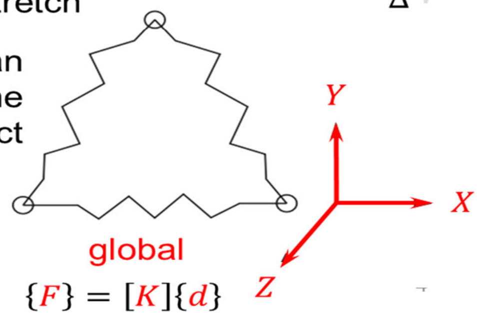

$k = \dfrac{F}{\Delta} (> 0)$ is the force needed to produce a unit stretch

The direct stiffness method: total stiffness matrix for an assemblage can be obtained by superimposing the stiffness matrices of the individual element in a direct manner




local

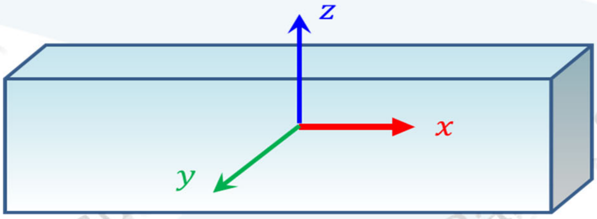
$\{f'\} = [k']\{d'\}$



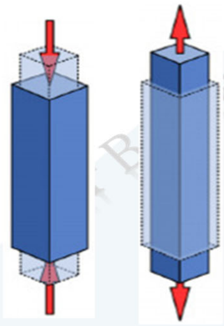

global

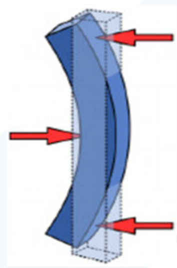
$\{F\} = [K]\{d\}$

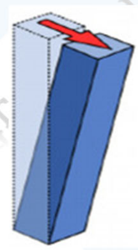
## *Internal forces*




| Axial (compression or tension) | Torsion | Bending | Shear |
|---|---|---|---|
| In direction of x | Around x | Around y or z | In direction of y or z |



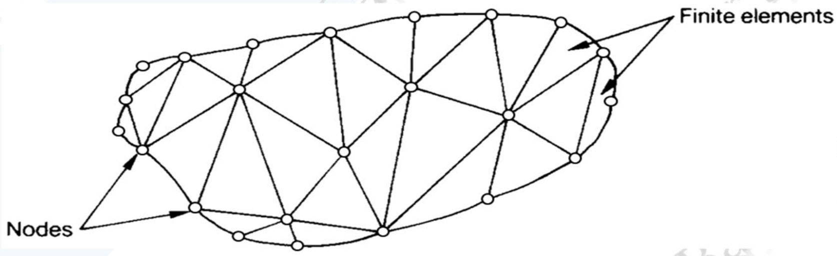




5

## What is the Finite Element Method?

FEM is based on representing a complex shape by a series of simpler shapes, where the simpler shapes are called finite elements. Ex. Lego (kids' play), Buildings.




Complex shape, represented by finite elements

The FEM is based on the <u>matrix displacement method</u>

Force (known) $\{\mathbf{F}\} = [\mathbf{K}]\{\mathbf{u}\}$ Displacement (unknown)

Stiffness (property of the element)

Finite Element Analysis is a numerical method to find <u>approximate solution</u> for displacements, stresses and strains that satisfy the differential form of Newton's equations of motion (done by utilizing <u>partial differential equations PDE</u>)

6

**Finite Element Method (FEM):** a numerical procedure for solving partial differential equations associated with field problems, with an approximate solution with an accuracy acceptable to engineers.

**Finite Element Analysis (FEA):** a discretized solution to a continuum problem using FEM.

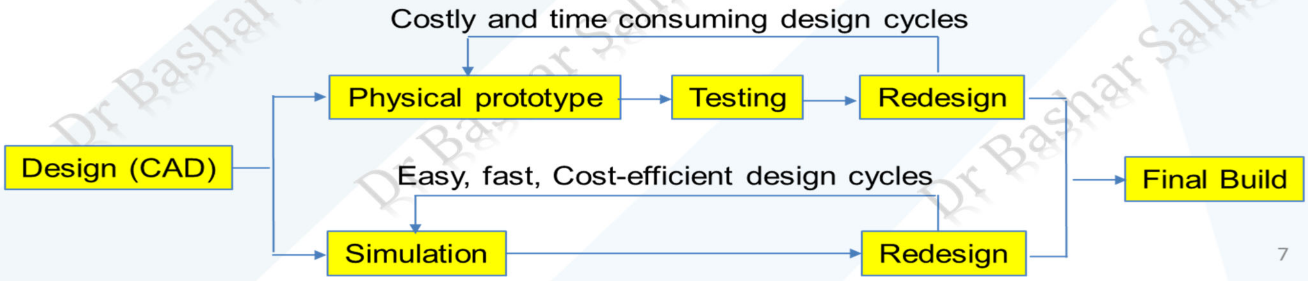
Actual structure ⇨ Assumptions, Equilibrium, Stress-strain law, compatibility, Virtual work ⇨ Structural model (FEM)

Partial differential equation can model nature (continuous)

System of linear equations (Algebraic equations) $\mathbf{F} = \mathbf{Ku}$

With today's computational power available, very large linear systems of equations can be solved (always an iterative process)

## How does the FEM help?




## Divide & Conquer

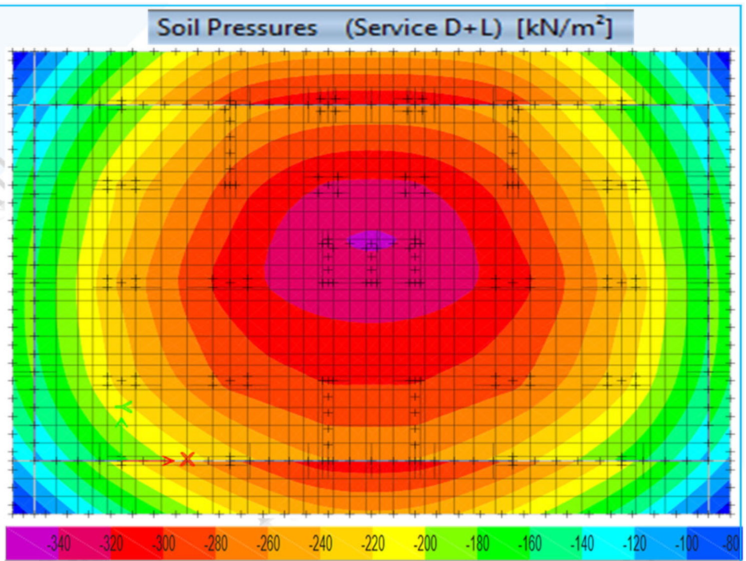
In FEM, a continuous domain is discretized into simple geometric shapes called elements.

A "finite element" is a small piece of structure

Nodes appear on the element boundaries and fasten the elements together

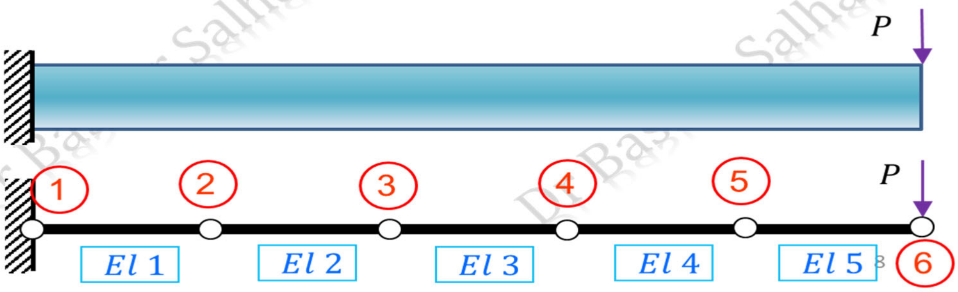
Continuum: infinite number of DoF

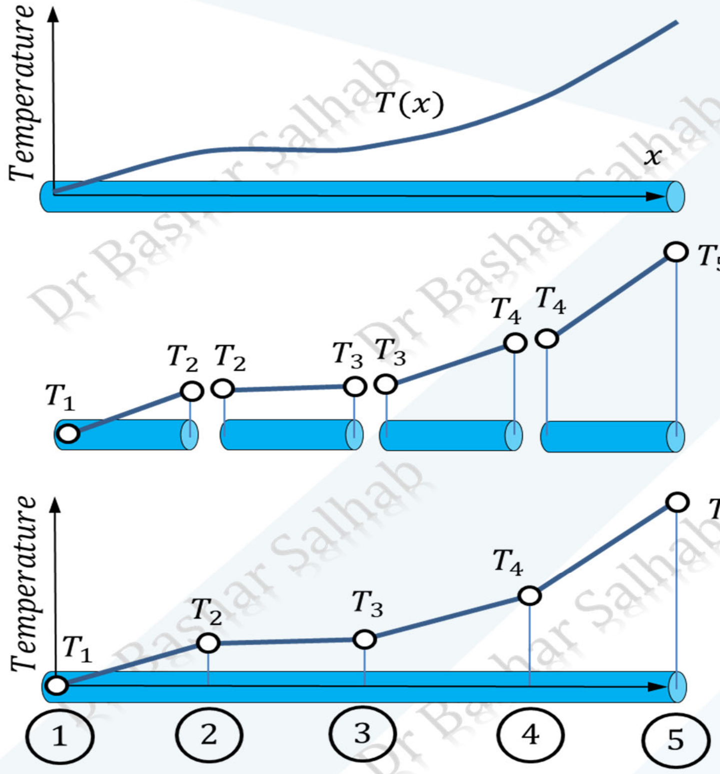
Discretized Model: finite number of DoFs




Structure

Finite element representation



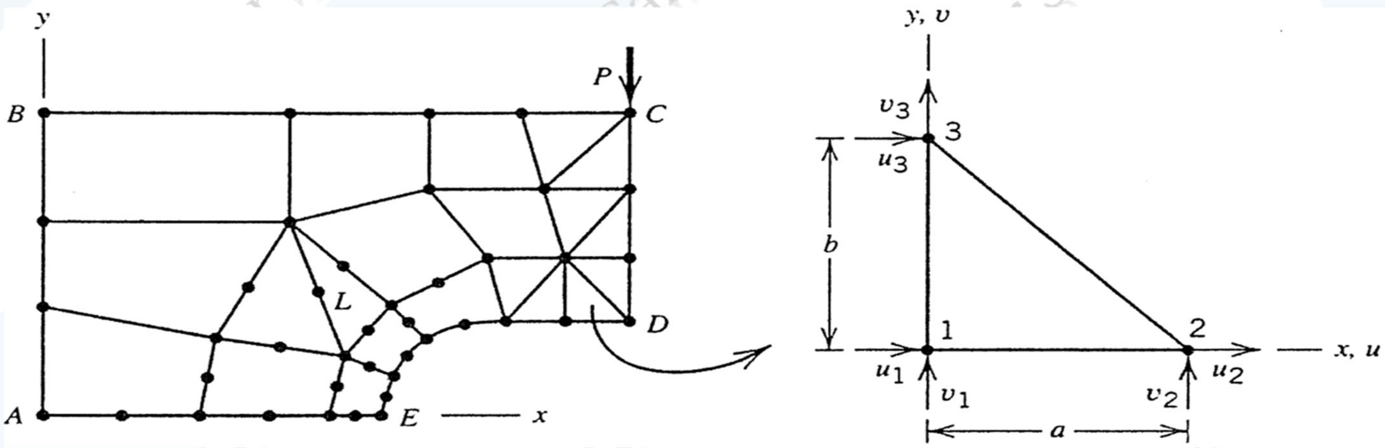



Using liner shape function. We can be more accurate by using 2nd order function

9

***First article on FEM:***

Turner, M. J., Clough, R. W., Martin H. C. and Topp, L. J. (1956), **"Stiffness and deflection analysis of complex structures"**, *Journal of the Aeronautical Science, Vol. 23 No. 9, pp. 805-823*




***This Course Main References:***

Kassimali A. (2011), **"Structural Analysis"**, *4th Edition SI Version, Cengage Learning, Stamford, U.S.A*

Logan D. L. (2022), **"A First Course in the Finite Element Method"**, *Enhanced 6th Edition SI Version, Cengage Learning, Boston, U.S.A*

10

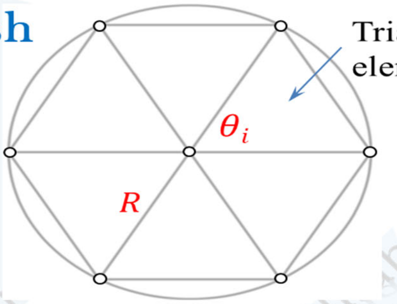
## Finite element mesh

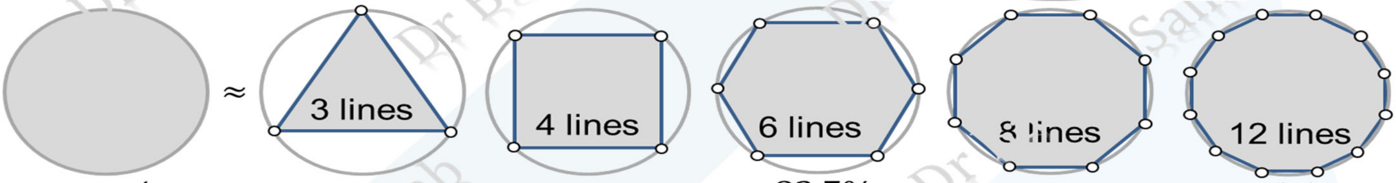
***Approximation of the area of a circle:***

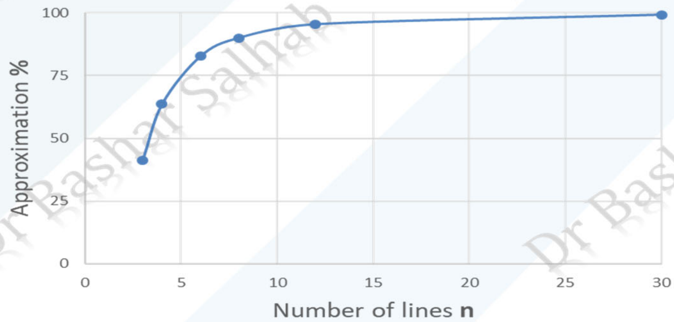
Area of 1 triangle: $E_i = 0.5R^2 \sin\theta_i$

Area of the circle $= \sum_{i=1}^{n} E_i = n\left(0.5R^2 \sin\frac{2\pi}{n}\right)$

$\lim_{n\to\infty} = \pi R^2$




11

***Discretization (mesh):***

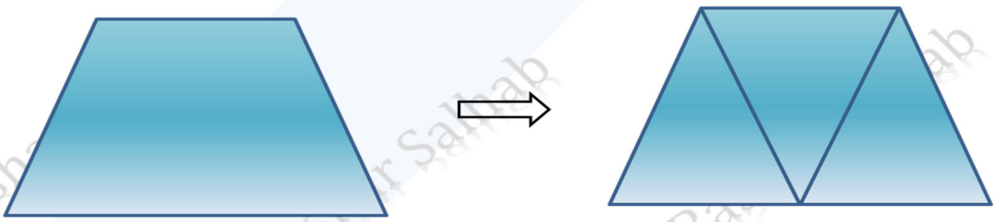
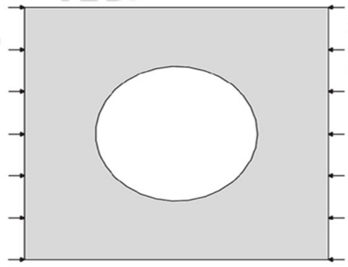
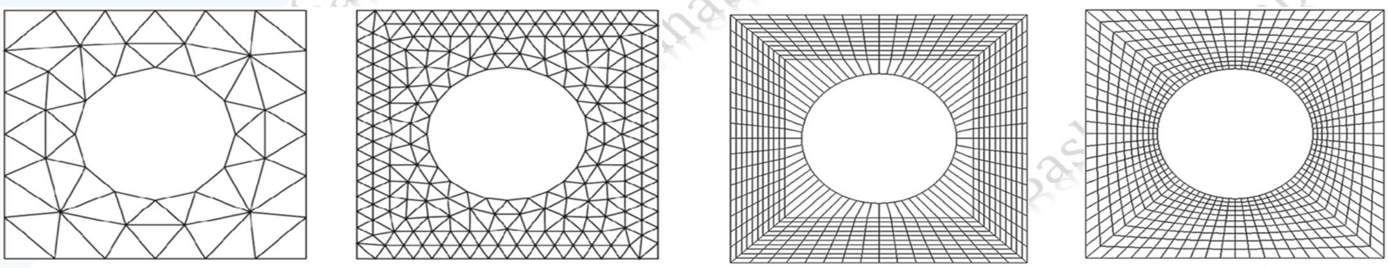


Plate with a hole

Triangular element



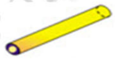
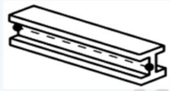


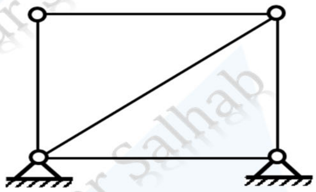

12

# Types of finite elements

**1D element (Line)** Connects two nodes in a straight line

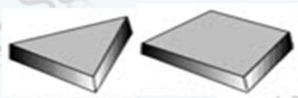
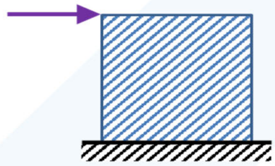
For long, slender structures with constant cross-sections

Actual models:  Rod (Truss) element ,  Beam element

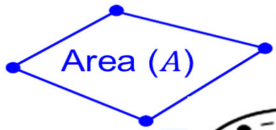


Finite element expressions (geometric properties defined by nodes)


**2D element (Plane)** Structural element with one small dimension and 2 large dimensions

Actual models:  Plates ,  Shear walls

Finite element expressions:




 Membrane element ,  Shell element

13

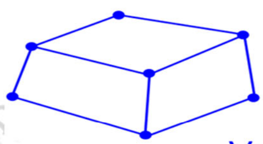
**3D element (Solid)** All geometric definitions included Volume elements, 3 translational DoF's

3-D fields (temperature, displacement, stress, velocity, etc.)

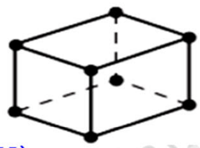
Actual models:

solid

Finite element expressions (geometric properties defined by nodes):

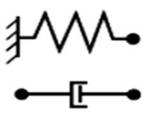


Volume $(V)$

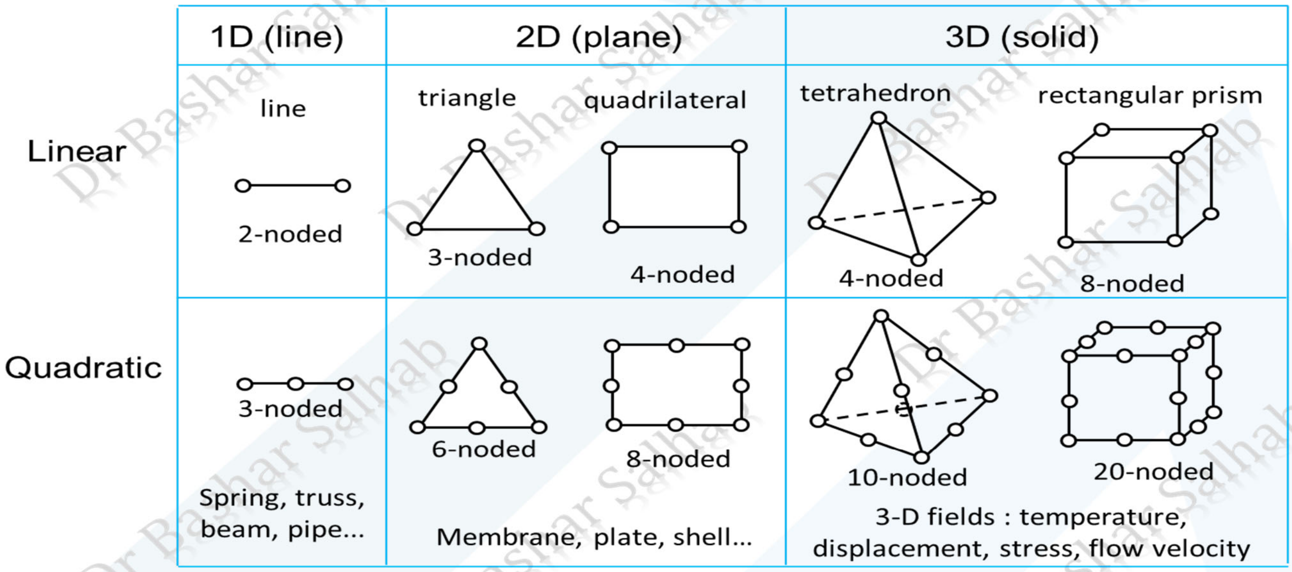


**Misc.** Connector element such as springs

Spring, Mass, Rigid link, etc

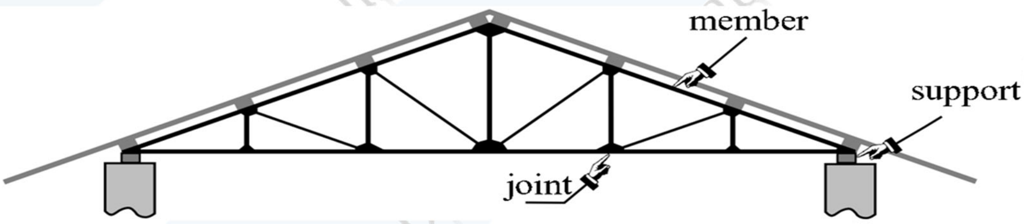


14



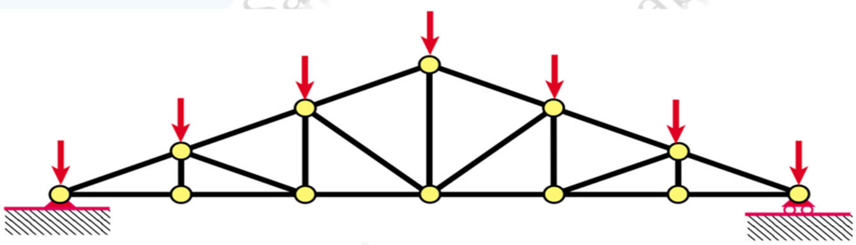

15

## Idealization

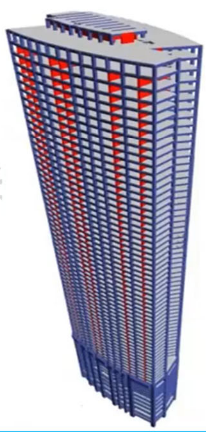


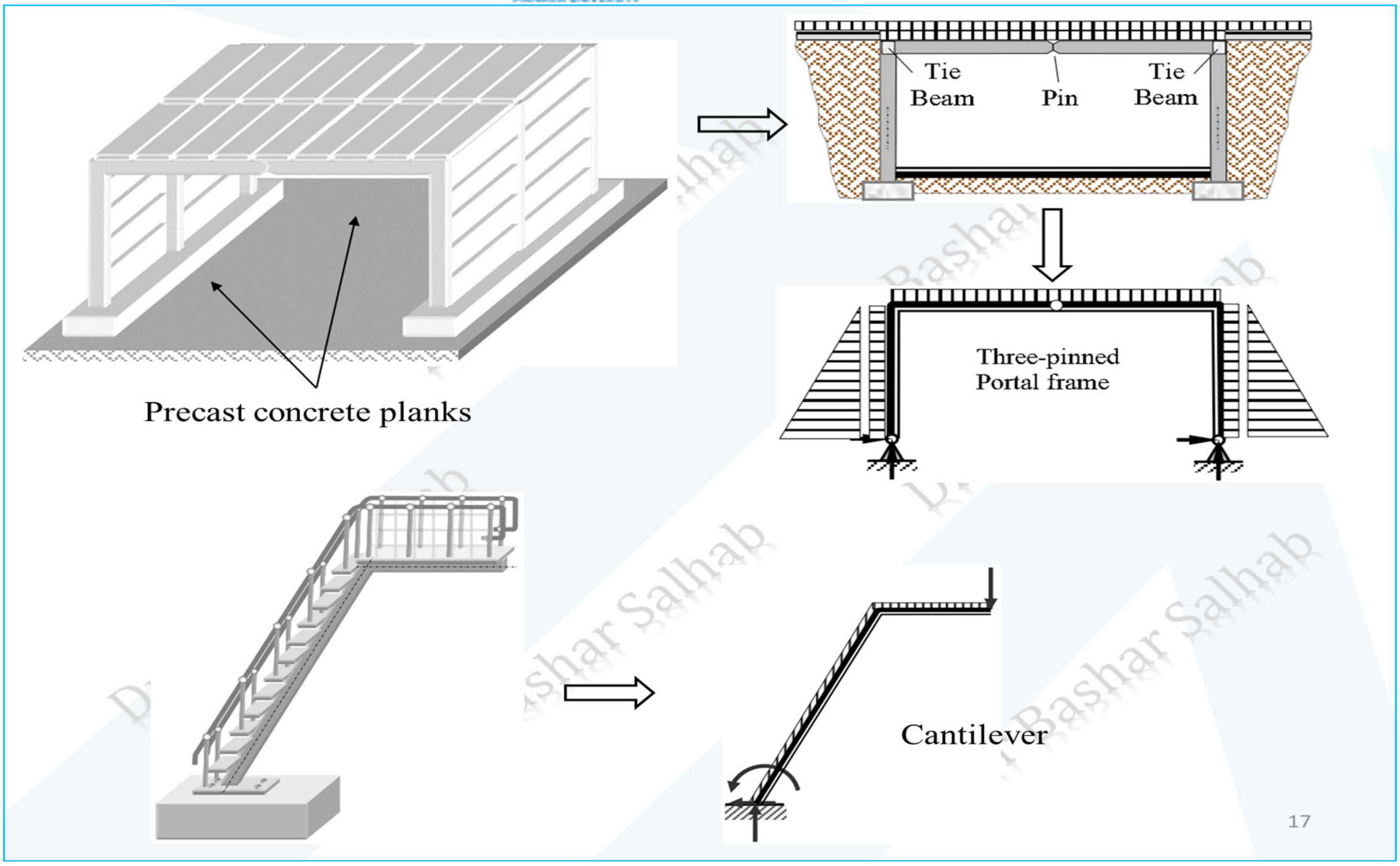

Physical System

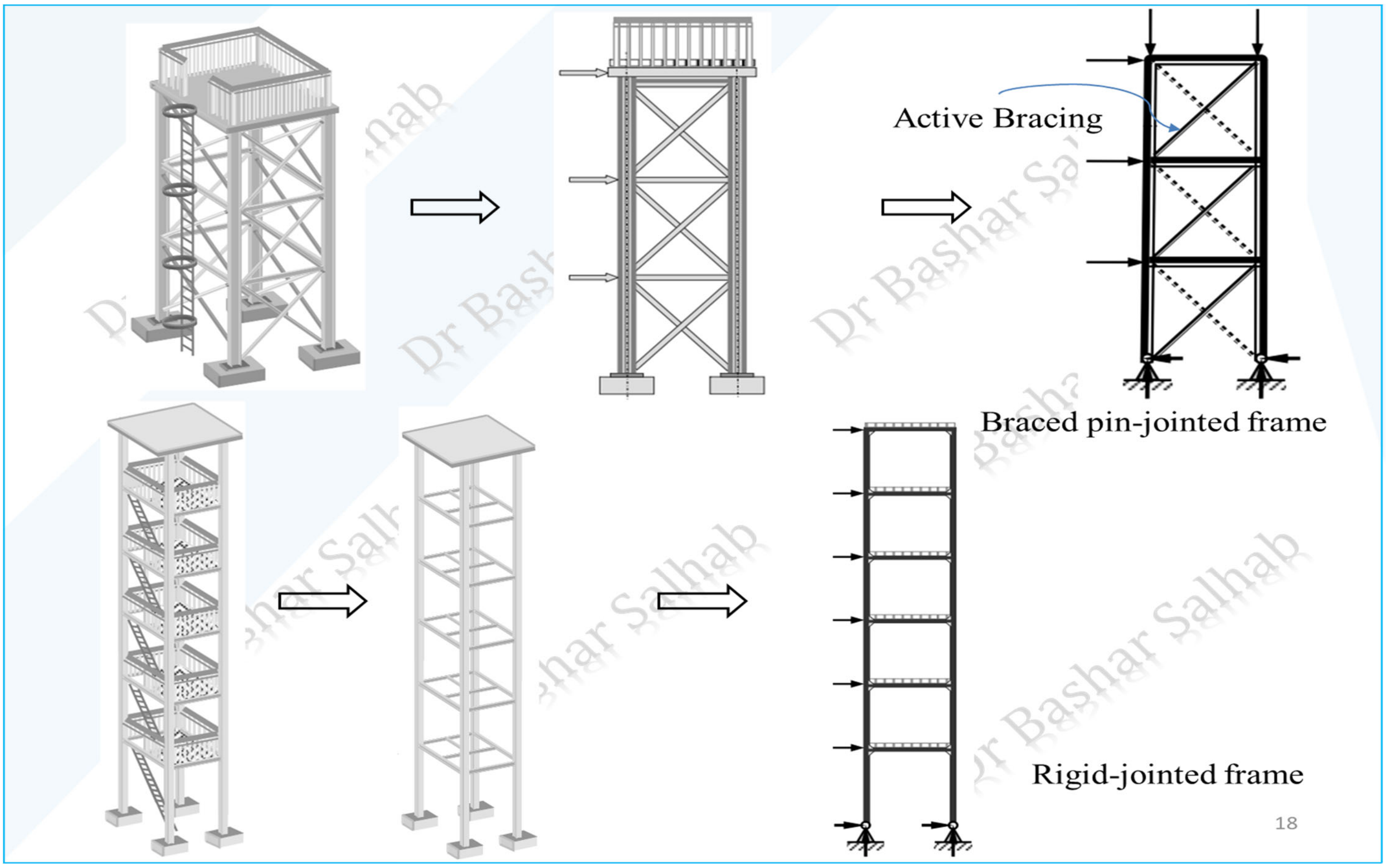
⇩

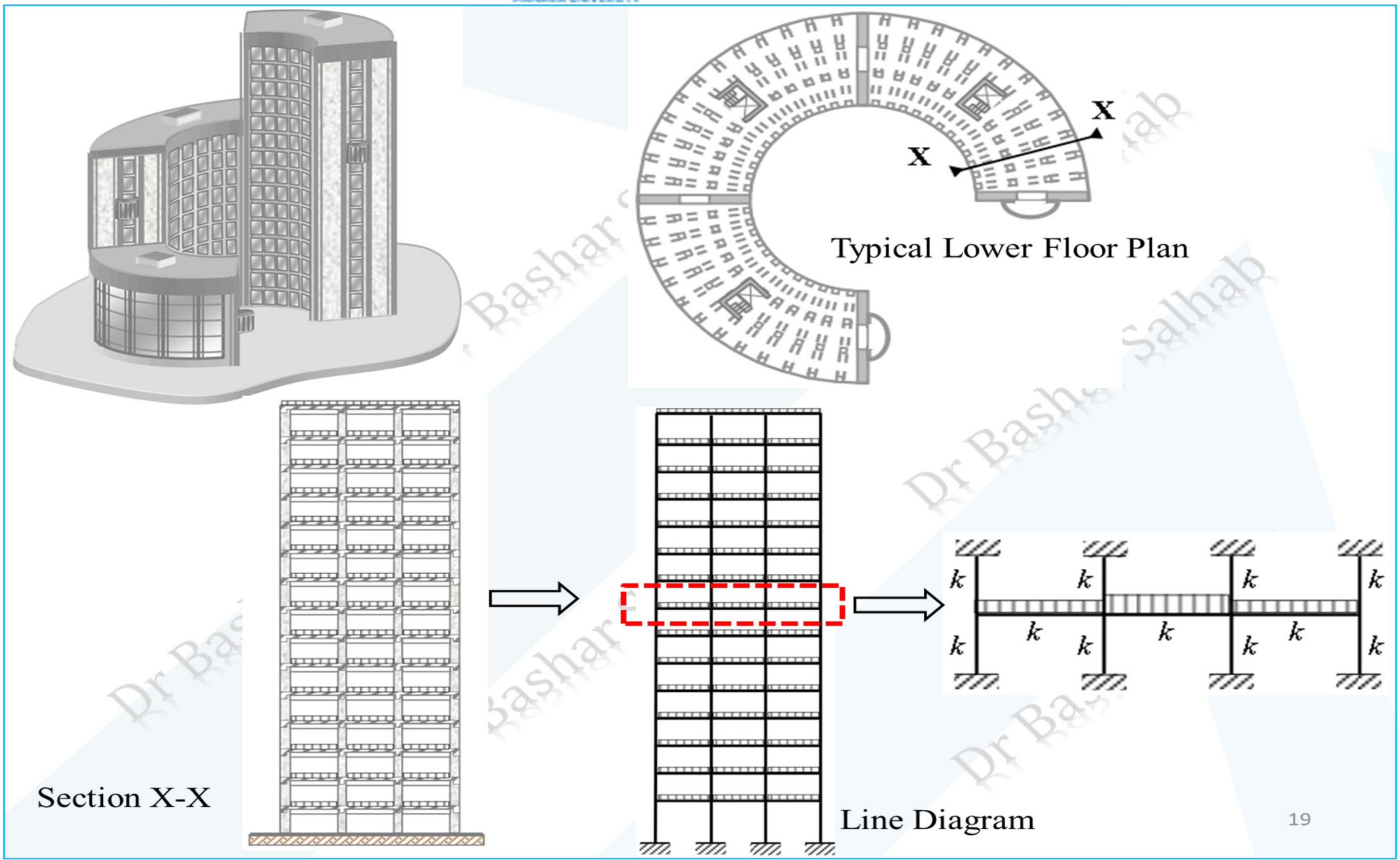


Structural model




Tie Beam
Pin
Tie Beam
Three-pinned Portal frame
Precast concrete planks
Cantilever
17


Active Bracing
Braced pin-jointed frame
Rigid-jointed frame
18




## Analytical Model

In the matrix stiffness method of analysis the structure is considered to be an assemblage of straight & prismatic members connected at their ends to joints.
A member is defined as a part of the structure for which the member force-displacement relations to be used in the analysis are valid.

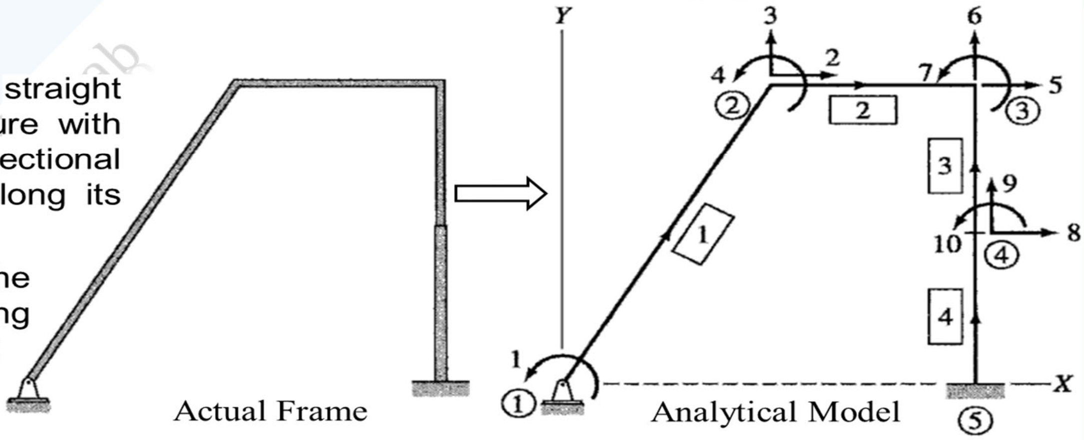
A joint is defined as a structural part of infinitesimal size to which the member ends are connected

Members & Joints are also referred as Elements & Nodes.

The model is a line diagram of the structure where joints & members are identified by numbers. Joint numbers are enclosed within circles, Member numbers are enclosed within rectangles.

A member is a straight part of the structure with constant cross-sectional properties $I$ & $A$ along its length.

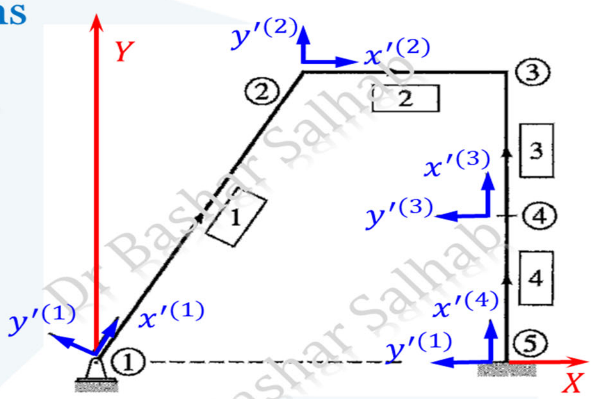
Arrow from the member beginning joint to its end joint



# Global & Local Coordinate Systems

The global (or structure) coordinate system $XYZ$ is a right-handed Cartesian or rectangular one starting at any arbitrary point.

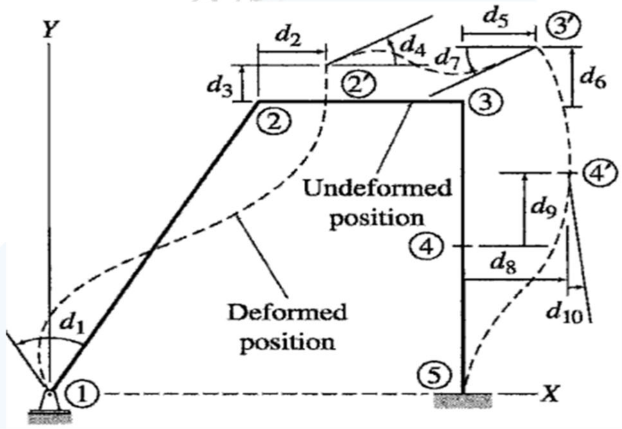
The local (or member) coordinate system $xyz$ , is a right-handed Cartesian coordinate system The origin of the local $xyz$ coordinate system for a member may be arbitrarily located at one of the ends of the member, with the $x$ axis directed along the centroidal axis of the member. The positive direction of the $y$ axis is chosen so that the coordinate system is right-handed, with the local $z$ axis pointing in the positive direction of the global $Z$ axis

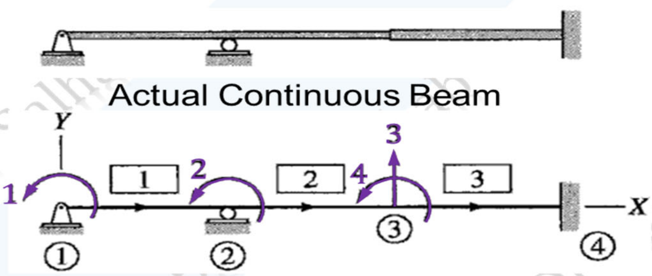



# Degrees of Freedom (DoF)

are the independent joint displacements (translations & rotations) that are necessary to specify the deformed shape of the structure when subjected to an arbitrary loading.

$$\mathbf{d} = \begin{Bmatrix} d_1 \\ d_2 \\ \vdots \\ d_9 \\ d_{10} \end{Bmatrix}$$

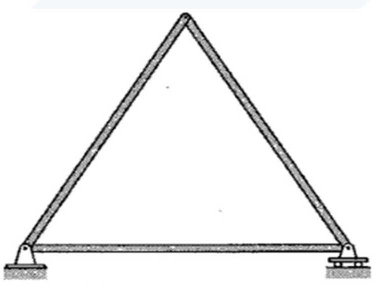


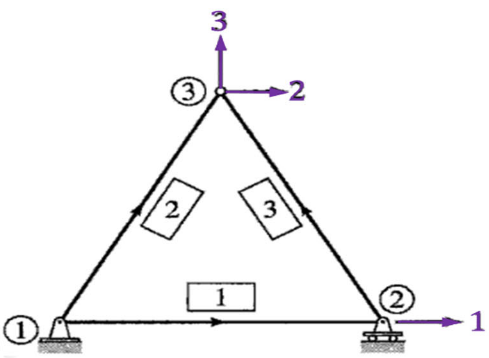

Actual Continuous Beam

Analytical Model and DoF

$$\mathbf{d} = \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \end{Bmatrix}$$



Actual Truss

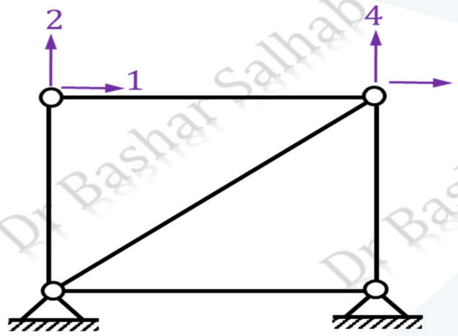



Analytical Model and DoF

$$\mathbf{d} = \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix}$$

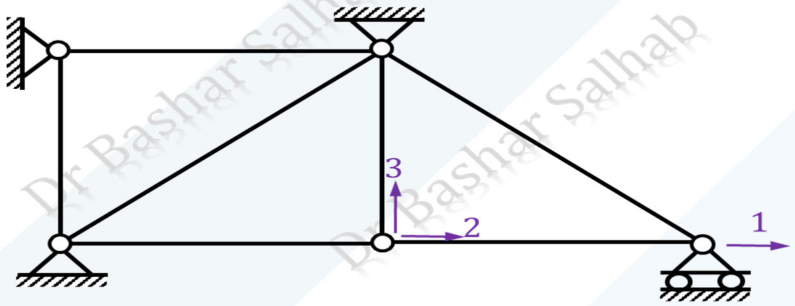
22

DoF for plane truss




$$\mathbf{d} = \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \end{Bmatrix}$$

Element number=5, DoF =4




$$\mathbf{d} = \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \end{Bmatrix}$$

23

## General steps of FEM:

Step 1: Discretize the continuum: Divide the body into an equivalent system of finite elements with associated nodes and choose the most appropriate element type.

Step 2: Choose a displacement function within each element.

Step 3: Find the element properties: Relate the stresses to the strains through the stress/strain law (generally called the constitutive law).

Step 4: Derive the stiffness matrix $[\mathbf{k}]$ for each element. Use the direct equilibrium method, a work or energy method, or a method of weighted residuals to relate the nodal forces to nodal displacements.

Step 5: Solve the global equation system: Assemble the element equations to obtain the $[\mathbf{K}]$ the stiffness matrix of the whole structure and introduce boundary conditions.

Step 6: Solve for the displacements at the nodes (unknown DoF) i.e solve simultaneous equations: $\{\boldsymbol{F}\} = [\mathbf{K}]\{\boldsymbol{u}\}$

Step 7: Solve for BMDs, SFDs, additional results (stress, strain, etc).

Step 8: Interpret and analyze the results for use in the design/analysis process.

24