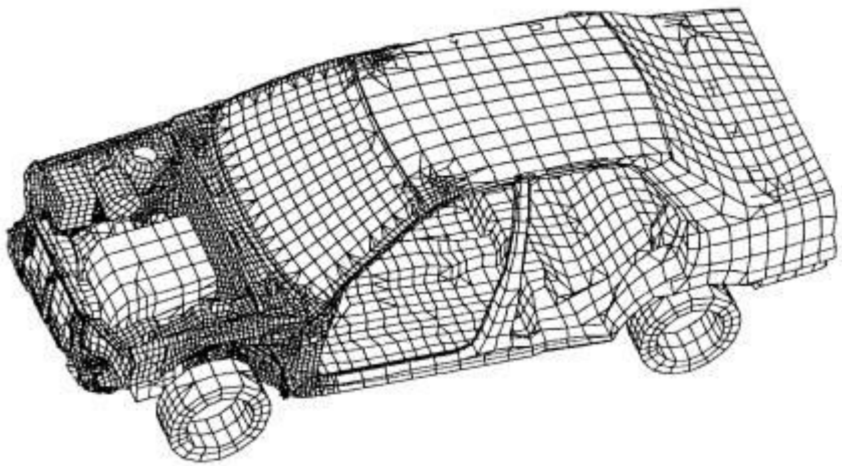
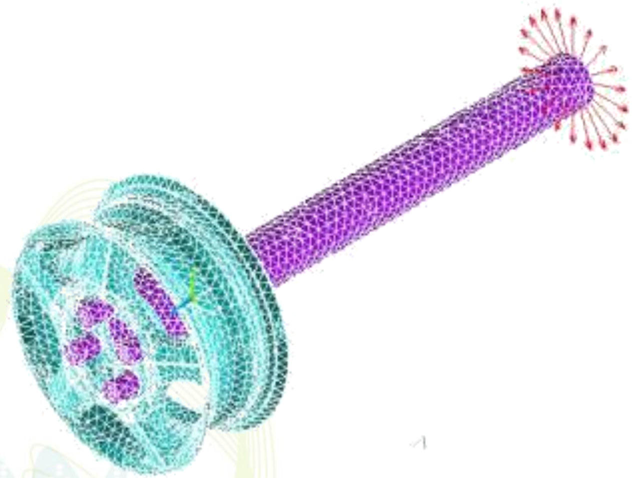
Chapter 9. Finite Element Analysis

- ✓ In practical mechanical design projects, the fundamental stress analysis equations are often not enough to handle the complicated geometry and boundary conditions.
- ✓ Design and analysis are no longer independent tasks. Finite Element Analysis (FEA) has become a fundamental design tool for design engineers. In many commercial software, finite element analysis has been combined with solid modeling to become integrated computer aided design or computer aided engineering software..
- ✓ Finite element analysis is a mathematical method that can be used to solve various problems such as stress analysis, heat conduction, electrical fields, magnetic fields, ideal fluid flow, etc.

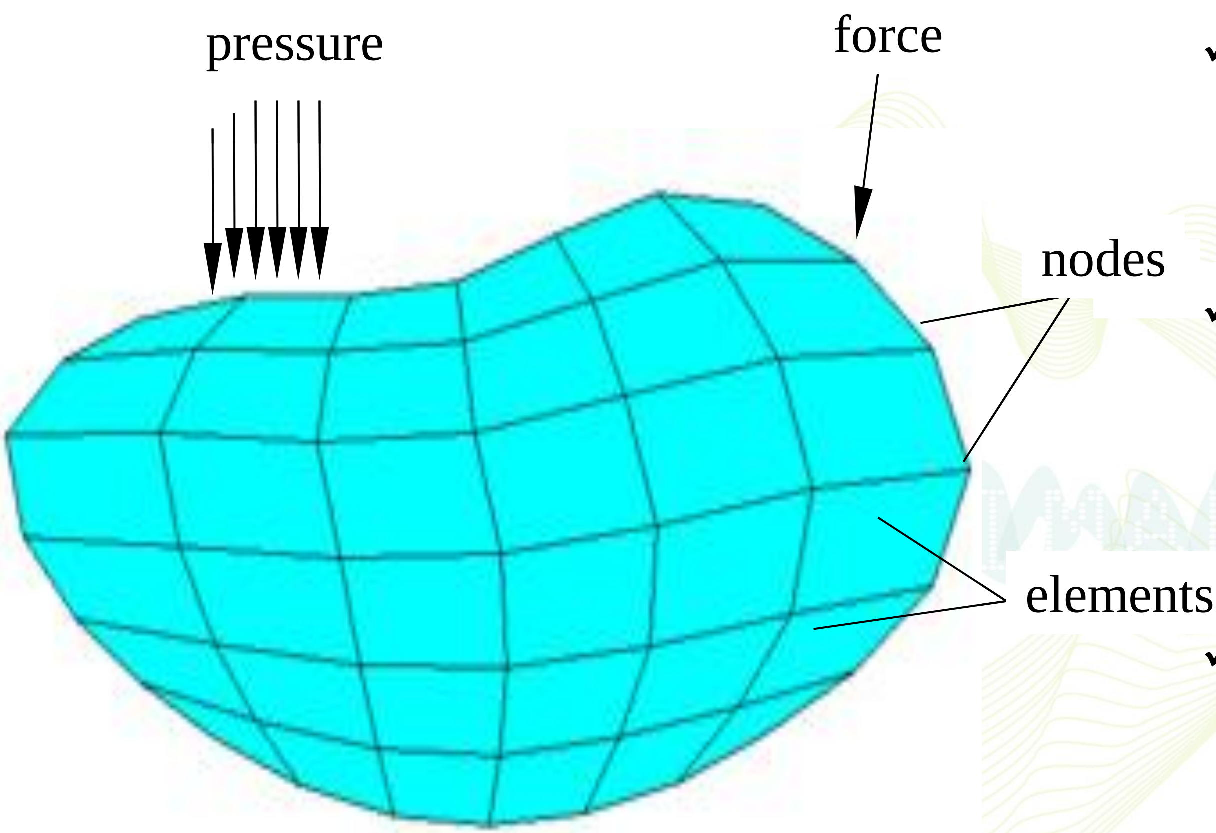
General Procedure for using an FEA Software





- ✓ In structural analysis, FEA can be use to solve for the displacement and stress for a complicated structure, under a given loading and boundary condition.
- ✓ Pre-processor → Solver → Post-processor
- ✓ Designers can define the geometry, loading, and boundary conditions in pre-processor.

Pre-Processor

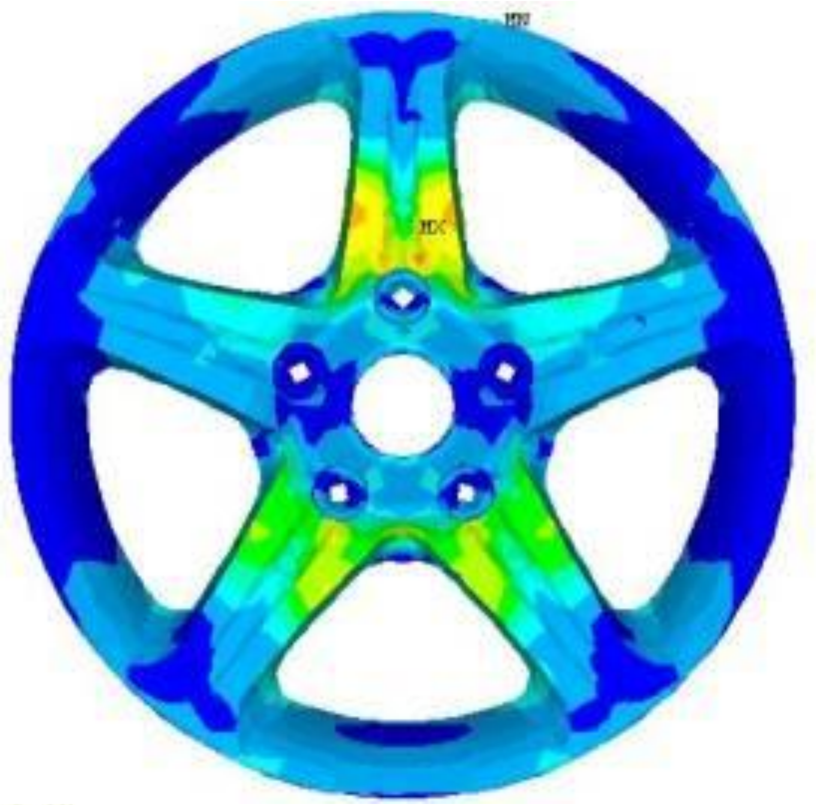


- ✓ In FEA method, an object is divided into a large number of elements, which is composed of nodes.
- ✓ Designers defines the geometry of the structure by defining the coordinates of nodes and connectivity of elements.
- ✓ Many FEA software provides automatic mesh generator.

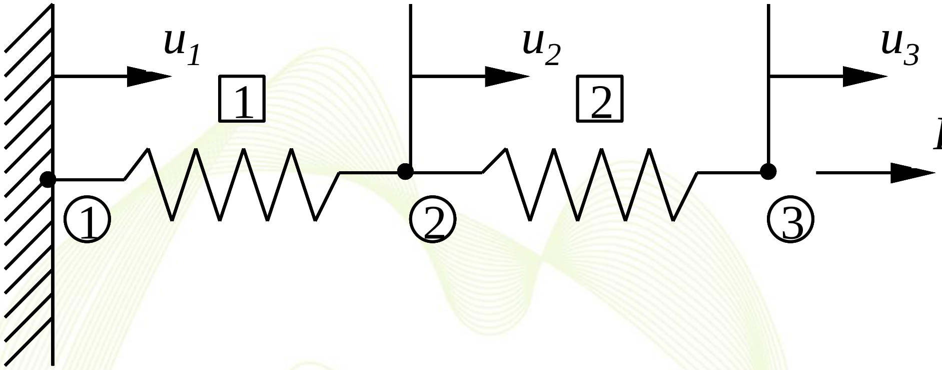
- ✓ In a finite element model, loading is often forces applied on nodes or pressure applied on the boundary of an element.
- ✓ Boundary conditions are represented by the degrees of freedom of the nodes.

Solver and Post-Processor

- ✓ In the solver, system equations are constructed according to the nodes, elements, loading, and boundary conditions defined by the designers. The system equations are then solved.
- ✓ The solution data is displayed graphically in the post processor to give the designer a more intuitive feeling.
- ✓ For example, nodal stresses are plotted in color, high stress in red, low stress in blue. Nodal displacement can also be enlarged and displayed graphically.

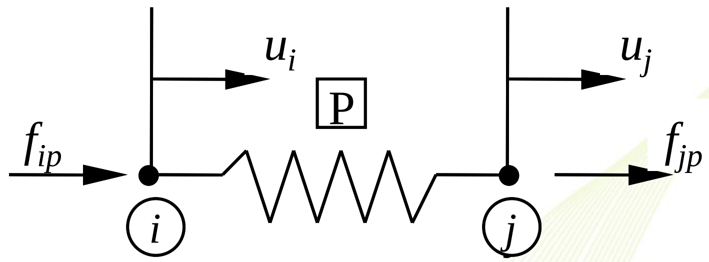


Building an FEA model -- One-Dimensional Spring



- ✓ Define coordinates of the 3 nodes.
- ✓ Choose the proper type of element considering required nodal degree of freedom from the element library. ➔There is only 1 DOF in each node in the spring elements used in the model.
- ✓ Assign material properties to the elements. ➔ Spring constant k_1, k_2.
- ✓ Define the connectivity of the elements. ➔ Geometry of the FEA model is defined by the coordinates of the nodes and connectivity of the elements.
- ✓ Define loading and boundary conditions. ➔ Horizontal force is applied on node 3, and node 1 is constrained.

A Typical Spring Element



$$\begin{bmatrix} k_p & -k_p \\ -k_p & k_p \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_{ip} \\ f_{jp} \end{Bmatrix} \qquad (5)$$

$$f_{ip} = k_p(u_i - u_j) \qquad (1)$$

$$f_{jp} = k_p(u_j - u_i) \qquad (2)$$

$$[\mathbf{k}]\{\mathbf{d}\} = \{\mathbf{f}\} \qquad (6)$$

Element stiffness matrix

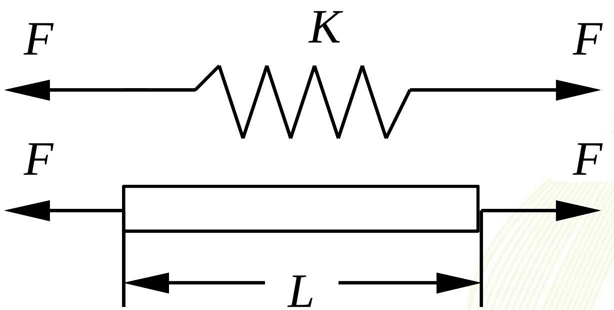
Force vector

$$f_{ip} = k_p u_i - k_p u_j \qquad (3)$$

$$f_{jp} = -k_p u_i + k_p u_j \qquad (4)$$

Displacement vector

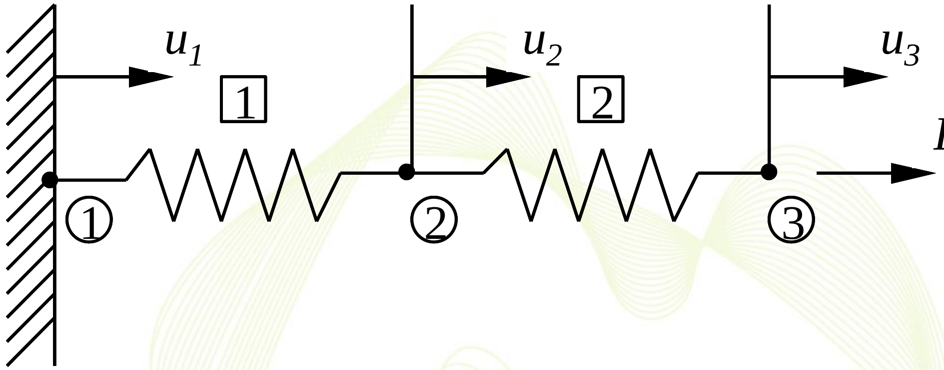
A Truss Element



$$[\mathbf{k}] = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix} \quad (11)$$

✓ The coordinate of the node (length L), element constant (cross sectional area A), material property (Young's modulus E) are all reflected in the element stiffness matrix.

Convert the FEA Model into System Equations (I)



✓ For element 1 and 2, Equation (5) becomes

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_{11} \\ f_{21} \end{Bmatrix} \quad (7)$$

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_{22} \\ f_{32} \end{Bmatrix} \quad (8)$$

Convert the FEA Model into System Equations (II)

Node 1: $\sum forces = 0 \Rightarrow f_{11} = F_1$

Node 2: $\sum forces = 0 \Rightarrow f_{21} + f_{22} = F_2$

Node 3: $\sum forces = 0 \Rightarrow f_{32} = F_3$

$$\begin{aligned} k_1u_1 - k_1u_2 \qquad\qquad\qquad\qquad &= F_1 \\ -k_1u_1 + k_1u_2 + k_2u_2 - k_2u_3 &= F_2 \\ -k_2u_2 + k_2u_3 \qquad\qquad\qquad &= F_3 \end{aligned}$$

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \qquad (9)$$

$$[\mathbf{K}]\{\mathbf{D}\} = \{\mathbf{F}\} \qquad (10)$$

K: system stiffness matrix or global stiffness matrix

Constructing the System Stiffness Matrix

Element 1

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \qquad \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 0 \\ F \end{Bmatrix} \tag{12}$$

Element 2

- ✓ In FEA software, components in the element stiffness matrices are "assembled" into proper positions in the system stiffness matrix.
- ✓ The connectivity of the elements are reflected in the system stiffness matrix.
- ✓ The boundary conditions are reflected in the displacement vector and the loading conditions are reflected in the force vector.

Solving the System Equations

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 0 \\ F \end{Bmatrix} \qquad (12)$$

$$\begin{bmatrix} k_1+k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ F \end{Bmatrix} \qquad (13)$$

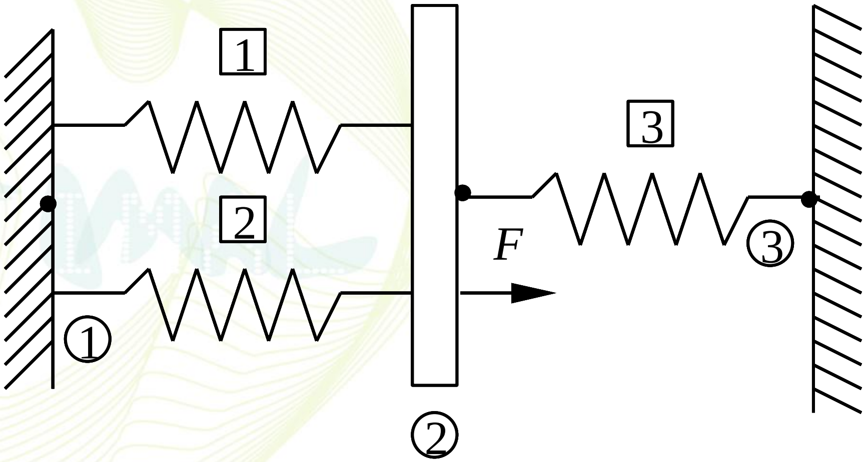
- ✓ In Equation (12), there are 3 linear equations and 3 unknowns. The degree of freedom of node 1 is constrained, therefore Equation (12) can be reduced to Equation (13).
- ✓ Equation (13) can be solved by Gaussian elimination, use forward elimination to convert the stiffness matrix into a triangular matrix, then use backward substitution to solve for the unknowns.
- ✓ Solving the system equations is the most computationally intensive process in finite element analysis.
- ✓ The size of the global stiffness matrix directly relates to the total number of degree of freedom in the finite element model.
- ✓ For a finite element model with large number of degree of freedom, the cost of computation and storage are high.

Example 1. Assemble and Solve the System Stiffness Matrix(I)

- ✓ Spring constants of the three springs are k_1=100N/m, k_2=200N/m and k_3=300N/m, respectively. The force applied is 150N. Find the displacement at node 2.

- ⇨ Construct the element stiffness matrix:

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



- ⇨ Assemble the system stiffness matrix

element1、element2

$$[\mathbf{K}] = \begin{bmatrix} k_1 + k_2 & -k_1 - k_2 & 0 \\ -k_1 - k_2 & k_1 + k_2 + k_3 & -k_3 \\ 0 & -k_3 & k_3 \end{bmatrix}$$

element3

Example 1. Assemble and Solve the System Stiffness Matrix(II)

$$[\mathbf{K}]\{\mathbf{D}\} = \{\mathbf{F}\}$$

$$\begin{bmatrix} k_1 + k_2 & -k_1 - k_2 & 0 \\ -k_1 - k_2 & k_1 + k_2 + k_3 & -k_3 \\ 0 & -k_3 & k_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \quad (14)$$

$$\begin{bmatrix} 300 & -300 & 0 \\ -300 & 600 & -300 \\ 0 & -300 & 300 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 150 \\ F_3 \end{Bmatrix}$$

$$600u_2 = 150 \Rightarrow u_2 = \frac{1}{4}(\text{m}) \qquad F_1 = F_3 = 75\text{N}$$

元智大學 最佳化設計實驗室
OPTIMAL DESIGN LABORATORY

Example 2. Calculating Stresses in a Truss Structure (I)

✓ A_1=A_2=10cm², A_3=5cm², A_4=4cm², F_2=500N, F_3=300N. Find the force and stress in each truss.

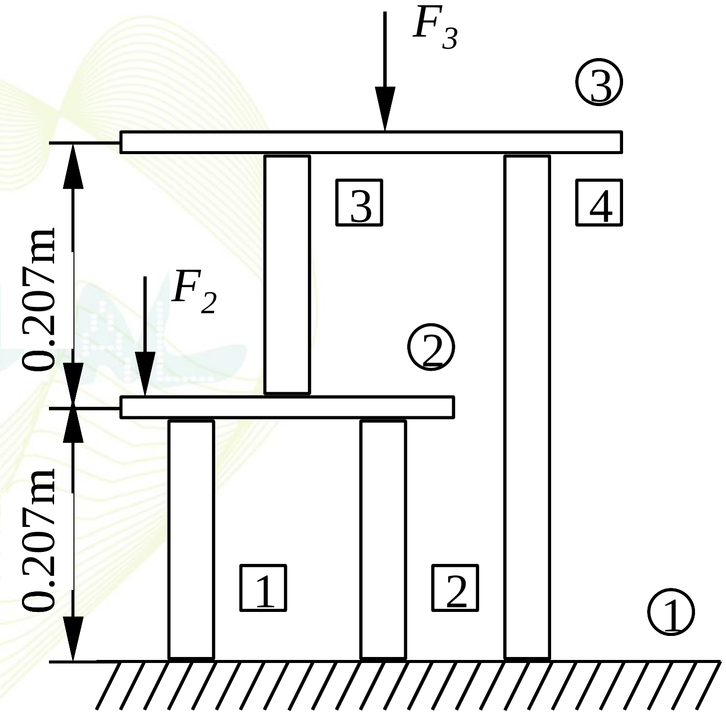
⇨ The element stiffness matrix:

$$[\mathbf{k}] = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

$$[\mathbf{k}_1] = [\mathbf{k}_2] = \begin{bmatrix} 1000 & -1000 \\ -1000 & 1000 \end{bmatrix} \times 10^6 \text{ N/m}$$

$$[\mathbf{k}_3] = \begin{bmatrix} 500 & -500 \\ -500 & 500 \end{bmatrix} \times 10^6 \text{ N/m}$$

$$[\mathbf{k}_4] = \begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix} \times 10^6 \text{ N/m}$$



Example 2. Calculating Stresses in a Truss Structure (II)

element1 、element2 — element4

$$[\mathbf{K}] = \begin{bmatrix} k_1 + k_2 + k_4 & -k_1 - k_2 & -k_4 \\ -k_1 - k_2 & k_1 + k_2 + k_3 & -k_3 \\ -k_4 & -k_3 & k_3 + k_4 \end{bmatrix}$$

element3 — element4

$$10^6 \times \begin{bmatrix} 2200 & -2000 & -200 \\ -2000 & 2500 & -500 \\ -200 & -500 & 700 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 500 \\ 300 \end{Bmatrix} \qquad (15)$$

$$u_2 = \frac{1}{3} \times 10^{-6}\,(\mathrm{m}) \qquad u_3 = \frac{2}{3} \times 10^{-6}\,(\mathrm{m}) \qquad F_1 = -800\,(\mathrm{N})$$

Example 2. Calculating Stresses in a Truss Structure (III)

- ✓ Strain of truss 1: $$\varepsilon_1 = \frac{(u_2 - u_1)}{L_1} = 1.61 \times 10^{-6}$$

- ✓ Stress of truss 1: $$\sigma_1 = E\varepsilon_1 = 0.333(\mathrm{MPa})$$

- ✓ Forces: $$f_1 = A_1 \times \sigma_1 = 333(\mathrm{N})$$

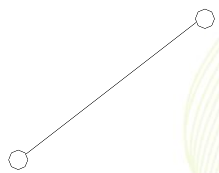
- ✓ Stress and forces produced in truss 3 and truss 4 are:
$$\sigma_3 = 0.333(\mathrm{MPa}) \quad f_3 = 166(\mathrm{N})$$
$$\sigma_4 = 0.333(\mathrm{MPa}) \quad f_4 = 133(\mathrm{N})$$

FEA Applications

- ✓ Messages in Solving Stage of FEA Software
  - Reading inputs
  - Computing element stiffness matrix
  - Assemble system stiffness matrix
  - Begin Gaussian elimination
  - Calculating displacements
  - Calculating stresses
- ✓ Analysis results can be displayed graphically in post-processing stage.
- ✓ Primary considerations when using FEA:
  - Do we really need to do a finite element analysis? Or a fundamental analysis is enough?
  - How much details do we need? Do we need a precise, quantitative analysis, or a conceptual, qualitative analysis?
  - How do we choose proper element types? How do we properly simulate the loading and boundary conditions?

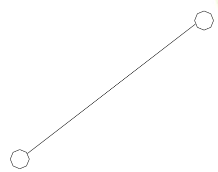
Common Element Types (I)

✓ The most important consideration when choosing element types is the degree of freedom of each node in that element.



truss element

- 2 nodes
- DOF of UX and UY
- Only axial load

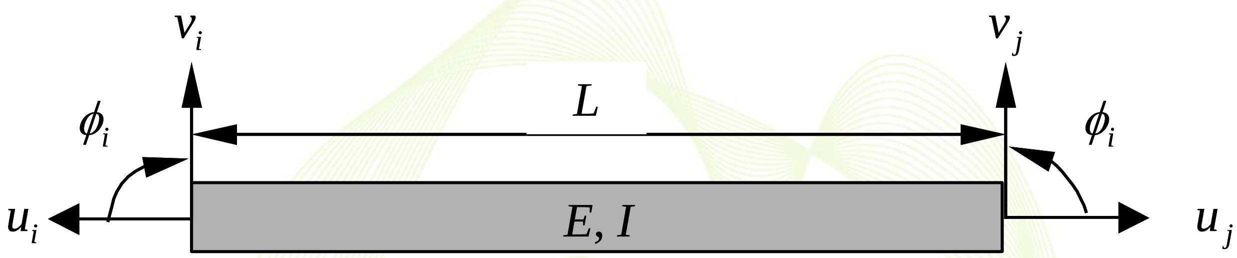


beam element

- 2 nodes
- DOF of UX and UY
- DOF of ROTZ
- Can sustain bending

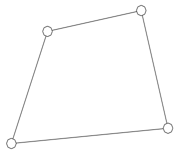
Beam Element

✓ In a 2-D beam element, there are three degree of freedom at each node. Therefore a 2-D beam element has a 6×6 element stiffness matrix.



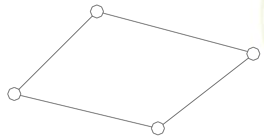
$$
\mathbf{k} = \begin{bmatrix}
\frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\
0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\
0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\
-\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\
0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\
0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L}
\end{bmatrix}
$$

Common Element Types (II)



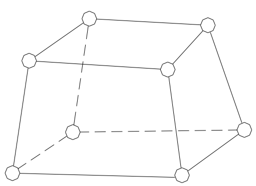
plane-stress element

- 4 nodes
- DOF of UX and UY
- For plane-stress and plane strain analysis



shell solid

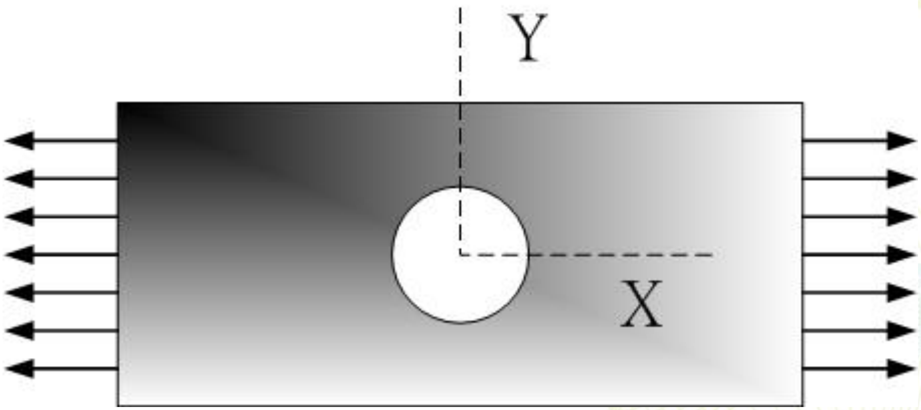
- 4 nodes
- DOF of UX, UY and UZ
- DOF of ROTX, ROTY and ROTZ
- For shell and plate analysis

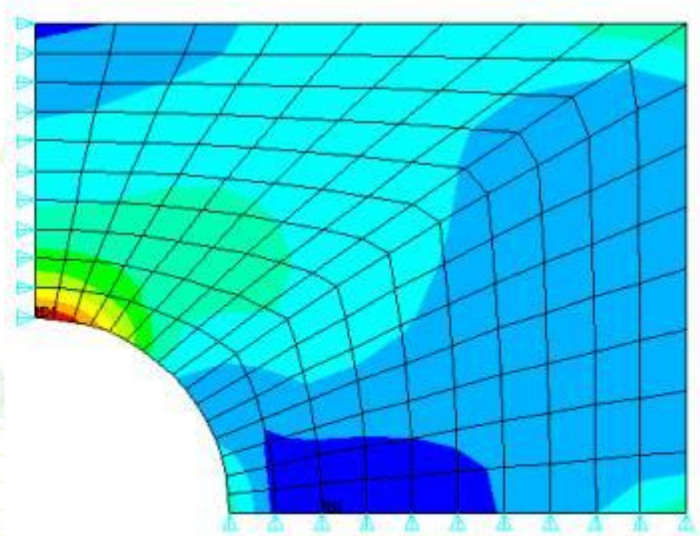


3D solid

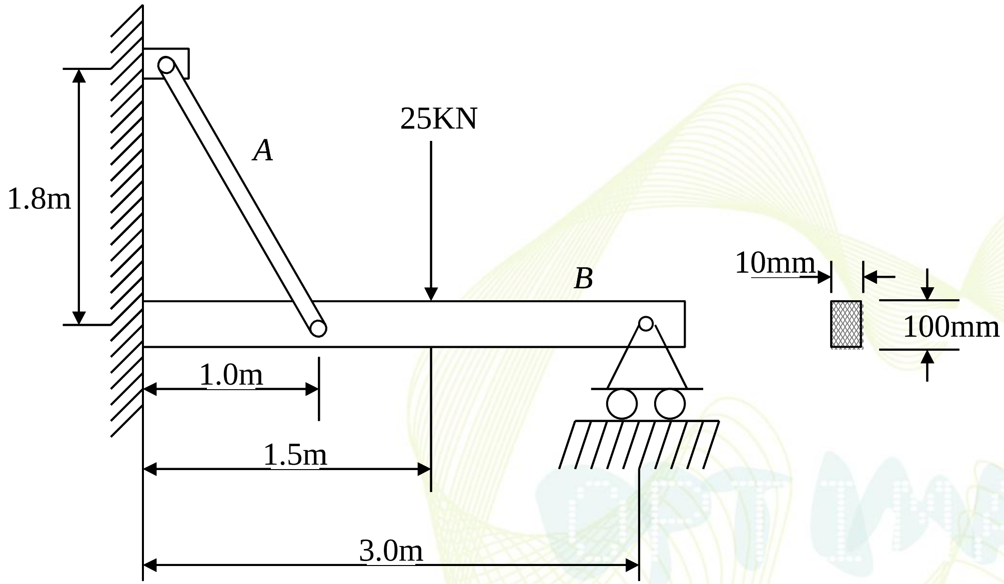
- 8 nodes
- DOF of UX, UY and UZ
- DOF of ROTX, ROTY and ROTZ
- For general analysis

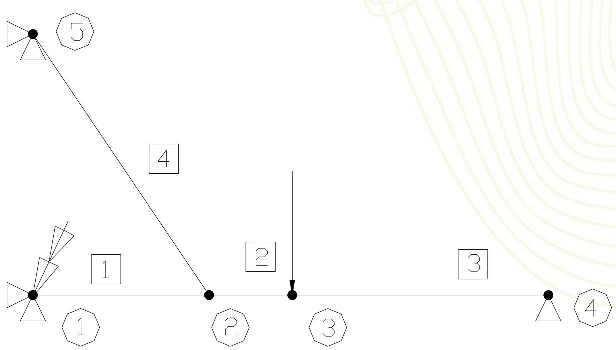
Example 3. Symmetric Boundary Conditions





Example 4. Planning an FEA Model

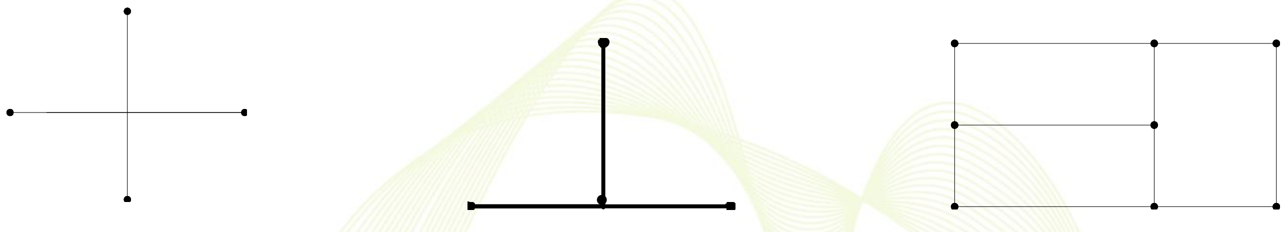




✓ Elements 1, 2, 3 are beam elements, element 4 is truss element.

Common Mistakes When Using FEA Software (I)

✓ Incorrect Connectivity

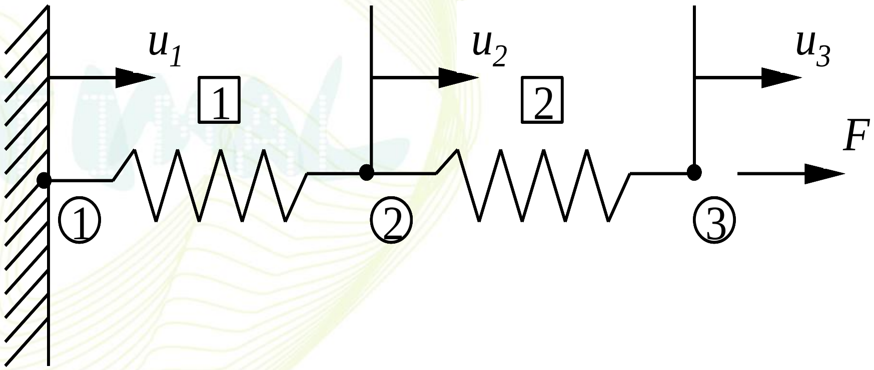


✓ Element distortion

Common Mistakes When Using FEA Software (II)

✓ Insufficient boundary conditions

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ F \end{Bmatrix} \qquad (16)$$



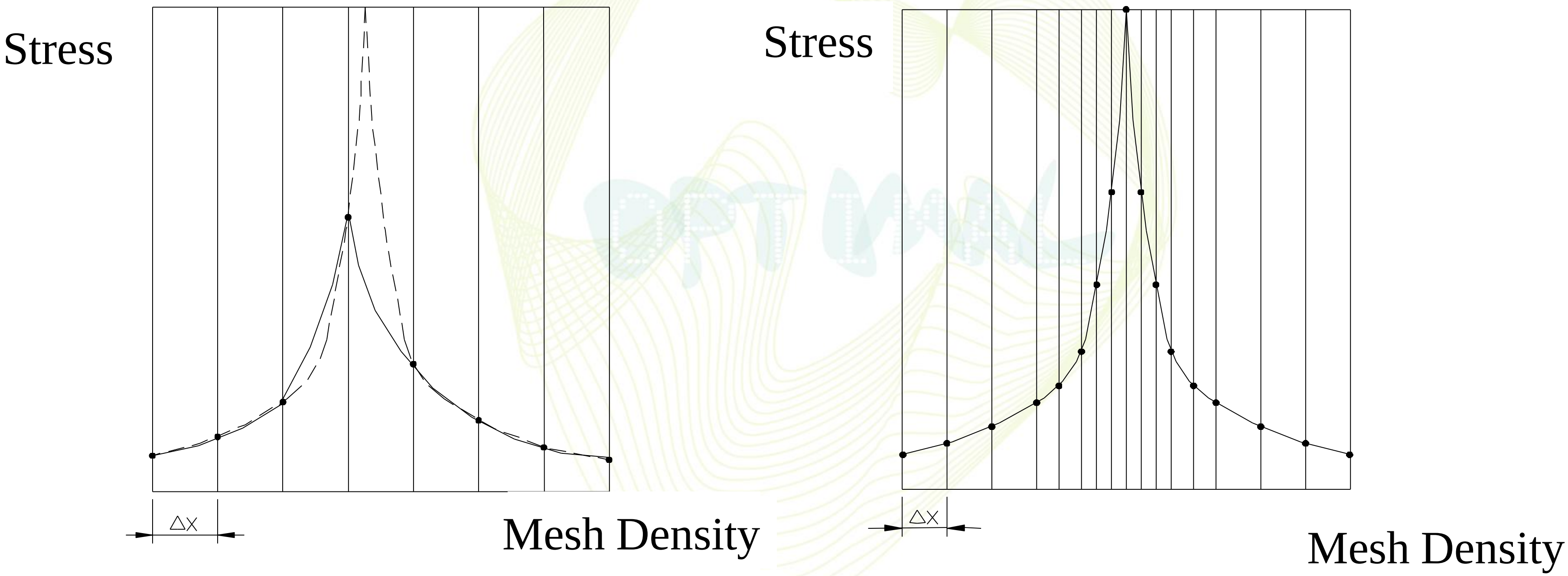
$$\begin{bmatrix} 1 & 0 & -1 \\ 0 & 1 & -1 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ F \end{Bmatrix} \qquad (17)$$

$$0 \cdot u_3 = F \qquad ??$$

Rigid-body motion would occur if DOF UX at node 1 exists.

Common Mistakes When Using FEA Software (III)

- ✓ Units have to be consistent.
- ✓ Use higher mesh density in the regions that have higher stress gradient.



元智大學 最佳化設計實驗室
OPTIMAL DESIGN LABORATORY

Example 5. Convergence Test to Decide Mesh Density