University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

## Computer Modelling Techniques

# FE-01-05

# SIMPLE 1D FINITE ELEMENTS

# Lecture Outline

# 2.1 Introduction

- The FE analysis of a simple **one-dimensional (1D) pin-jointed element** is described and later extended to cover an assembly of 1D elements.
- These pin-jointed structures consist of **long thin elements** linked together by **frictionless pin-joints**, which are assumed to transmit only axial forces to the elements.

*Important*

- These elements **do not bend**. Elements that allow bending are referred to as "beam elements" which are more complex than pin-jointed elements.
- Pin-jointed members are also referred to as **"*trusses*".** Trusses are loaded only at the joints and the weight of the members may be neglected.

## 2.2 A Simple Uniaxial 1D Pin-Jointed Element

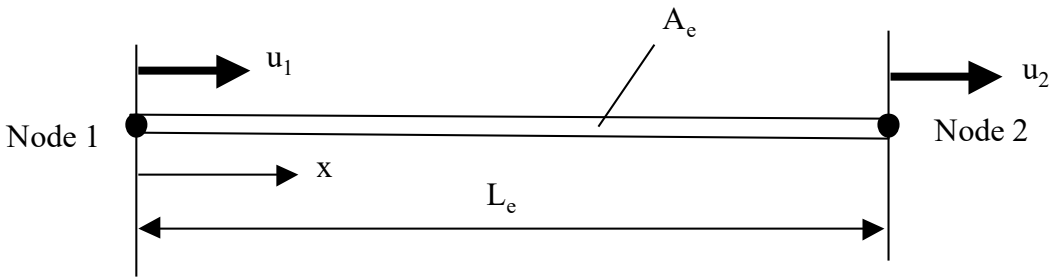


*Figure 1: A one-dimensional bar element*

- The main strategy in formulating the FE equations is to derive an expression for the element '*stiffness*', i.e. treating the element as if it were a spring of stiffness $k$, as follows:

$$k\ u = F$$

- where $k$ is the stiffness, $u$ is the displacement and $F$ is the force.

- Assuming that the material is linear elastic, the uniaxial stress-strain relationship is given by Young's modulus as follows:

$$\sigma = E\ \varepsilon$$

- For a uniaxial bar, the strain is defined as the change in length divided by the original length, as follows:

$$\varepsilon = \frac{\Delta L}{L_e} = \frac{u_1 - u_2}{L_e}$$

- Note that this definition of strain is simplistic, and only applies to a uniaxial long bar under tension or compression. In 2D and 3D continuum problems, a more sophisticated definition of strain must be used.

The stress in the bar is given by:

$$\sigma = \frac{F}{A_e}$$

Substituting the stress and strain results in:

$$\frac{F}{A_e E} = \frac{u_1 - u_2}{L_e}$$

Hence, a general force-displacement relationship can be obtained as follows:

$$F = \frac{A_e E}{L_e}\left(u_1 - u_2\right)$$

At nodes 1 and 2, the forces can therefore be expressed as two simultaneous equations, as follows:

$$\frac{A_e E}{L_e}(u_1 - u_2) = F_1$$

$$\frac{A_e E}{L_e}(u_2 - u_1) = F_2$$

which can be expressed in matrix form as follows:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

where $k_1 = \frac{E A_e}{L_e}$

The matrix expression can be expressed in a more concise way as follows:

$$[k_e][u_e] = [F_e]$$

***Important***

where $[k_e]$ is the "*element stiffness matrix*" (of size 2x2), $[u_e]$ and $[F_e]$ are the element displacement and force "vectors", respectively (each of size 2x1).

**Note:** The stiffness matrix is a **symmetrical matrix**. The symmetry is fortunate as it saves a considerable amount of computational time when using a numerical equation solving algorithm.

## 2.3 A More General Energy Derivation of Element Stiffness

- In more sophisticated elements, such as 3D continuum elements or shell elements, using an **equilibrium approach is very tedious** and is not adaptable to other elements.
- For a more generalised approach that is applicable to all element geometries, **the energy formulation is widely used** to derive FE formulations.
- The derivation of the FE formulation for any element of any geometry can be usually broken down into **7 main steps**.

## Step 1: Define the element and the shape (interpolation) functions

- A uniaxial element is said to have one ***"degree of freedom"*** per node, i.e. only one independent variable (the uniaxial displacement).

***Important***

- The displacement, ***u*, is always used as the independent variable** in FE formulation, as it automatically satisfies the compatibility equations, i.e. the element faces move together with no gaps or overlaps.

- If forces are used as the independent variables, the elements will not be compatible.

For a 2-node element, we assume a linear displacement function of $x$ as follows:

$$u = C_1 + C_2\, x$$

where $C_1$ and $C_2$ are constants.

These constants can be expressed in terms of the nodal displacements ($u_1$ and $u_2$) by satisfying the element's boundary conditions, which are:

- At node 1 (where $x = 0$), $u = u_1$
- At node 2 (where $x = L_e$), $u = u_2$.

Therefore, the constants $C_1$ and $C_2$ can be expressed in terms of $u_1$ and $u_2$ as follows:

$$u_1 = C_1 + 0 \quad \Rightarrow \quad C_1 = u_1$$

$$u_2 = C_1 + C_2\, L_e \quad \Rightarrow \quad C_2 = \frac{u_2 - u_1}{L_e}$$

The displacement function can therefore be expressed in terms of the nodal displacements as follows:

$$u = u_1 + \left( \frac{u_2 - u_1}{L_e} \right) x$$

which can be rearranged as follows:

$$u = \left( 1 - \frac{x}{L_e} \right) u_1 + \left( \frac{x}{L_e} \right) u_2$$

- This process is **similar to curve-fitting** where a straight line equation is obtained from the coordinates of two points at either end.

- Using $u_1$ **and** $u_2$ **(instead of** $C_1$ **and** $C_2$**) is physically more meaningful** since it expresses the displacement of any point on the element as a function of the two displacements at the corner nodes.

- The functions that multiply the nodal displacements $u_1$ and $u_2$ are called the ***"shape functions"*** **or the "*interpolation functions*".**

- For a 2-node element, the shape functions are linear.

- If more nodes are used per element, e.g. three nodes, the expression for the displacement function and the shape functions become quadratic.

## Step 2: Satisfy the material law (constitutive equations)

The material law in this simple uniaxial problem is simply given by Young's modulus definition, as follows:

$$\sigma = E\,\varepsilon$$

## Step 3: Derive the element stiffness matrix

- This energy approach is much **more convenient than the force equilibrium approach**, particularly when more complex 3D elements are used.

- The principle of **minimum total potential energy (T.P.E.)** can be used to minimise the strain energy function with respect to the nodal displacements.

- The principle states that the TPE must be minimised with respect to the displacements. The TPE is expressed as follows:

$$T.P.E. = U - W$$

where
$U$ is the strain energy
$W$ is the work done by the forces

The **strain energy, *U*, is given by:**

$$U = \frac{1}{2}[\sigma][\varepsilon] \times Volume$$

Therefore, for a uniaxial element of length $L_e$, the strain energy is:

$$U = \int_0^{L_e} \frac{1}{2}\,\sigma\,\varepsilon\,(A\,dx)$$

Using the more accurate definition of strain as a differential of the displacement function, the strain in this one-dimensional element is given by:

$$\varepsilon = \frac{du}{dx}$$

Since

$$u = \left(1 - \frac{x}{L_e}\right)u_1 + \left(\frac{x}{L_e}\right)u_2$$

The strain can be expressed as follows:

$$\frac{du_x}{dx} = \left(\frac{-1}{L_e}\right)u_1 + \left(\frac{1}{L_e}\right)u_2 = \frac{u_2 - u_1}{L_e}$$

Substituting for stress in terms of strain, the strain energy expression can be expressed as a function of the displacement as follows:

$$U = \int_0^{L_e} \frac{1}{2} (E\,\varepsilon)(\varepsilon)\, A\, dx$$

$$= \int_0^{L_e} \frac{1}{2} E\, (\varepsilon)^2\, A\, dx$$

$$= \frac{E\,A}{2} \int_0^{L_e} \left( \frac{u_2 - u_1}{L_e} \right)^2 dx$$

$$= \frac{E\,A}{2\,L_e{}^2} (u_2 - u_1)^2 \left[ x \right]_0^{L_e}$$

$$= \frac{E\,A}{2\,L_e} \left( u_2{}^2 - 2\,u_2\,u_1 + u_1{}^2 \right)$$

The **work done by the forces $F_1$ and $F_2$** is simply expressed as the force multiplying the displacement at each node, as follows:

$$W = F_1 u_1 + F_2 u_2$$

Therefore, the **T.P.E. expression** can be written in terms of the displacements as follows:

$$T.P.E = \frac{E\,A}{2\,L_e}\left(u_2{}^2 - 2u_2\,u_1 + u_1{}^2\right) - \left(F_1\,u_1 + F_2\,u_2\right)$$

Since there are two displacements in the TPE expression, **the principle of minimum TPE requires minimisation with respect to both $u_1$ and $u_2$ .** This yields two equations.

Minimising TPE **with respect to $u_1$** gives:

$$\frac{\partial T.P.E.}{\partial u_1} = 0 = \frac{E\,A}{2\,L_e}\left(-2\,u_2 + 2\,u_1\right) - F_1$$

Similarly, minimising T.P.E. **with respect to $u_2$** gives:

$$\frac{\partial T.P.E.}{\partial u_2} = 0 = \frac{E\,A}{2\,L_e}\left(2\,u_2 - 2\,u_1\right) - F_2$$

The above two equations can be rearranged as follows:

$$0 = \frac{E\,A}{L_e}\left(u_1 - u_2\right) - F_1$$

$$0 = \frac{E\,A}{L_e}\left(-u_1 + u_2\right) - F_2$$

which can be combined in matrix form as follows:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

where $k_1 = EA/L_e$ , which is **identical to the expression derived using the force.**

## 2.4 Element Assembly (More than one element)

All FE formulations start with the derivation of the stiffness matrix for a single element, and then combining the element with its neighbouring elements (element assembly).

Two important relationships must be satisfied in the element assembly:

(i) The **displacement of a particular node must be the same** for every element connected to it.

(ii) The externally applied forces at the nodes on the surface must be **balanced by the 'internal' forces** on the elements at the nodes.

## Step 4: Assemble the overall stiffness matrix

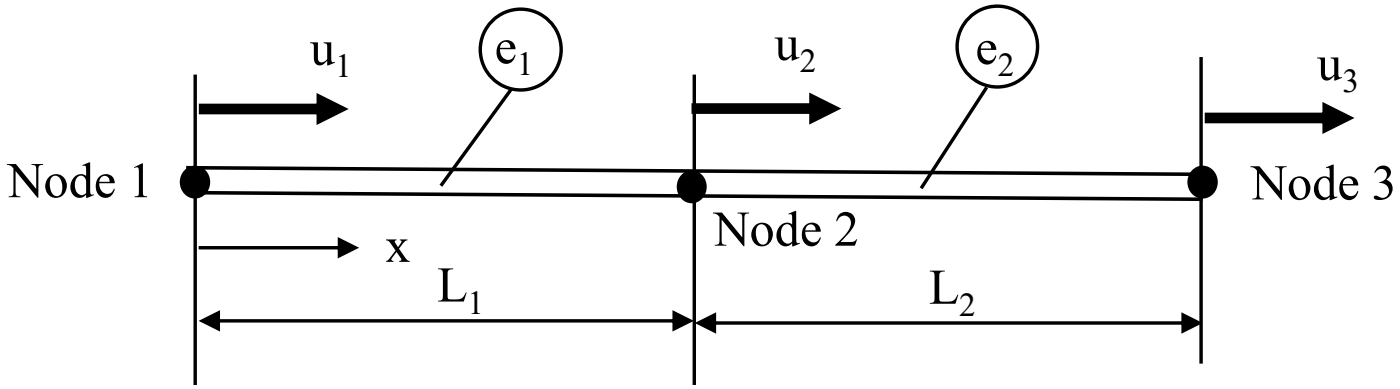


*Figure 2: An element assembly of two uniaxial pin-jointed elements*

For generality, we assume that the **elements have different lengths**, $L_1$ and $L_2$, and different stiffnesses, $k_1$ and $k_2$.

Taking **each element in turn**, the individual element stiffness matrices can be constructed and then assembled together.

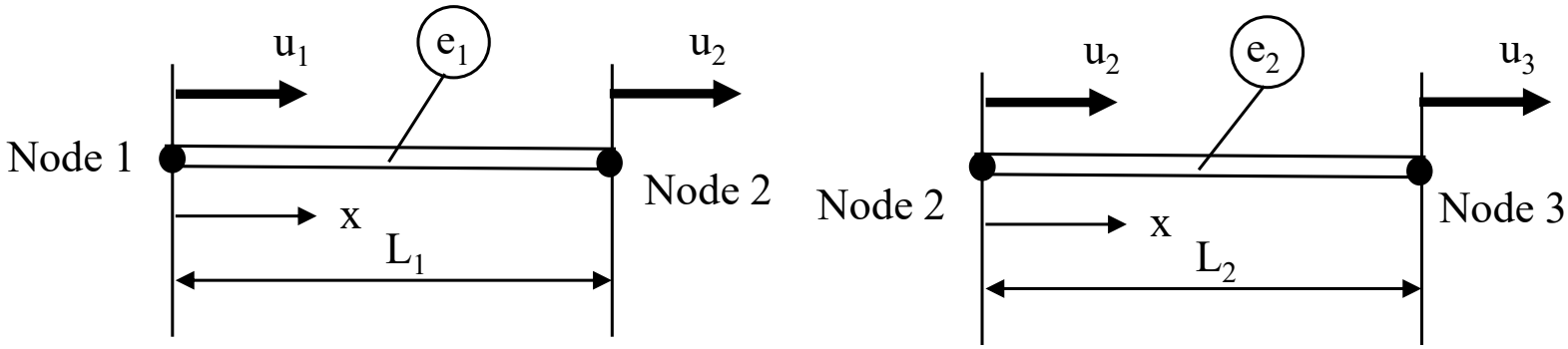


*Figure 3: Individual elements in the assembly*

For element $e_1$, the 'internal' force on node 1 is given by:

$$F_1 = k_1 \left( u_1 - u_2 \right)$$

Similarly, for element $e_2$, the 'internal' force on node 3 is given by:

$$F_3 = k_2 \left( u_3 - u_2 \right)$$

Since node 2 is shared by the two elements, the 'internal' $F_2$ can be written as two components:

$$\left(F_2\right)_{e_1} = -F_1 = -k_1 \left( u_1 - u_2 \right)$$
$$\left(F_2\right)_{e2} = -F_3 = -k_2 \left( u_3 - u_2 \right)$$

Therefore, the total 'internal' force on node 2 is given by:

$$F_2 = \left(F_2\right)_{e_1} + \left(F_2\right)_{e_2} = -F_1 - F_3 = -k_1 \left( u_1 - u_2 \right) - k_2 \left( u_3 - u_2 \right)$$

Therefore, three simultaneous equations can be written for $u_1$, $u_2$ and $u_3$, as follows:

$$F_1 = k_1 u_1 - k_1 u_2$$
$$F_2 = -k_1 u_1 + (k_1 + k_2) u_2 - k_2 u_3$$
$$F_3 = -k_2 u_2 + k_2 u_3$$

The above 3 equations can be assembled in matrix form as follows:

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

which can be written as a general equation for the stiffness of the assembly $[K]_{Assembly}$ (the whole FE mesh) as follows:

$$[K]_{Assembly} \; [u]_{Assembly} = [F]_{Assembly}$$

Again, note the **symmetry of the assembly stiffness matrix**.

## Step 5: Apply the boundary conditions and external loads

To obtain a **'unique' solution** of the problem, some displacement constraints (called 'boundary conditions)' and some loading (force) conditions must be prescribed at some of the nodes.

These conditions usually take one of the following forms:

(i) **Prescribed displacement (called 'Boundary Condition'):**
A zero or non-zero prescribed nodal displacement, or sliding against a rigid surface.

(ii) **Prescribed load (force):**
An applied force in a given direction or a prescribed pressure.

Note that if a force is not prescribed at a given node, it is automatically assumed to have a prescribed nodal force of zero (i.e. a free surface)..

***Important***

As an example, consider the problem shown below.

**(i) A prescribed displacement (i.e. boundary condition) of $u_1=0$.**

Note that both $u_2$ and $u_3$ are unknown and will be calculated by solving the simultaneous equations.

**(ii) A non-zero prescribed external force (load) of $F_3=W$.**

Note that nodes 1 and 2 are automatically assigned a zero prescribed external force, i.e. $F_1=0$ and $F_2=0$.

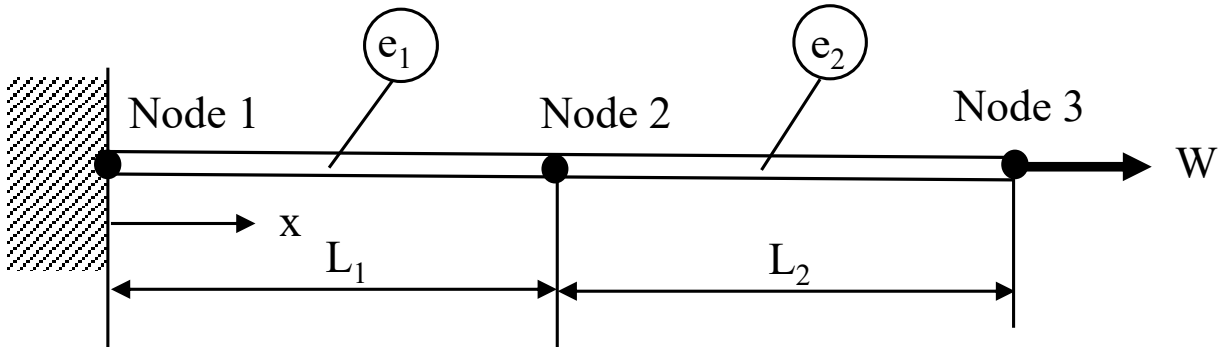


*Figure 4: Element assembly example with boundary conditions and loads*

Implementing the prescribed values in the assembly equation gives:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{bmatrix} 0 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ W \end{bmatrix}$$

Since $u_1$ is given, the first equation is not required

$$\begin{bmatrix} k_1+k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 0 \\ W \end{bmatrix}$$

## Step 6: Solve the simultaneous equations

- Most FE practical applications contain thousands or hundreds of thousands of nodes, and can only be **solved numerically**.
- Standard numerical equation solvers, such as the **Gaussian elimination technique** can be used to solve the equations.
- By solving the simultaneous equation by algebraic manipulation, the two unknown displacements can be obtained:

$$u_2 = \frac{W}{k_1}$$

$$u_3 = \frac{W}{k_2} + \frac{W}{k_1}$$

## Step 7: Compute other variables

- After solving the assembly equations, **displacements at all the nodal points** are determined.
- From the displacement values, the **element strains** can be obtained from the strain-displacement relationship.
- The **element stresses** are obtained from the material law.

*Important*

- Remember that **only the displacements are used as the independent variables**.
- For this reason, the computed **FE displacements are usually (slightly) more accurate that the computed FE stresses**.

*Important*

## 2.5 Summary of Key Points

- In deriving the FE formulation, the first step is to **define the order of variation** (e.g. linear or quadratic) of displacement (not force) over each element.

- **Displacements are the only 'independent' variables** in FE formulations. All other variables (such as force, stress, strain, etc.) are derived from the displacements.

- A **linear shape function** can be used with 2-node pin-jointed elements.
  This means that the displacement is allowed to vary linearly per element

  The **strain, which is a differential of the displacement, is therefore constant per element**.

  For linear elastic analysis, the **stress is also constant per element** since it is linearly dependent on strain.

- For each element, **an element stiffness expression** can be derived as follows:

$$[k_e][u_e]=[F_e]$$

- Two alternative approaches can be used to derive the element stiffness matrix; either a **direct equilibrium (non-energy) approach**, or a **more general energy approach** (minimising the total potential energy (TPE)).

- The individual element stiffness matrices are **assembled together in the global stiffness matrix** by combining the forces (and element stiffness) of the nodes which are shared between two or more elements.

- The element stiffness matrix and the overall stiffness matrix of the assembly are always **symmetric**.

- To obtain a unique solution of the simultaneous equations, **boundary conditions and loads** must be prescribed at some of the nodes.