Chapter 3

Major Steps in Finite Element Analysis

Instructor

Tai-Ran Hsu, Professor
Department of Mechanical Engineering
San Jose State University
San Jose, California
Chapter Learning Objectives

1) Lean the basic principle of finite element analysis with “discretization” of continuum media into finite number of “elements” interconnected at “nodes.”

2) Learn the formulation of finite element analysis, including the derivations of “element equations” and “overall stiffness equations” by variational process developed by Rayleigh-Ritz and Galerkin principles.

3) Learn the derivation of Interpolation function relating the “element quantities” with corresponding “nodal quantities”

4) Major steps involved in general finite element analysis
Steps in the Finite Element Method

FEM is now used in a wide cross-section of engineering analyses

It is not possible to establish a set of standard procedure for all the computations for all problems

Following will be a general guideline.

Most FEM follow eight (8) steps
We have learned the reasons for discretizing a real continuum or structure into a finite number of ELEMENTS interconnected at NODES. There are 7 different shapes of elements for discretization:

1. **Bar elements** for modeling trusses, frames and beams

2,3) **Plate elements** for plate type structures, or with physical phenomena varying on plans. Quadrilateral plate element can be treated as assembly of 2 or 4 triangular plate elements

4,5) **Torus elements** for structures of axisymmetric geometry with radius such as cylinders, tubes, disks, and cylindrical structures with various radii along the length. Quadrilateral torus element can be treated as assemblies of 2 or 4 triangular torus elements

6,7) **Tetrahedron and Hexahedron elements** for modeling 3-dimensional continua or structures. Hexahedron elements can be treated as assembly of 3 or 5 tetrahedron elements.

- It is common to use **hybrid element shapes** for modeling same continua or structures
- Most commercial FE codes have “**automatic mesh generator**” together with built-in element library, as will be described in a latter part of this course.
Establish the FE mesh with set coordinates, element numbers and node numbers

- The discretized FE model must be situated with a coordinate system, for example:
  1) the x-coordinate for bar elements
  2) the x-y coordinates for plate elements
  3) the r-z coordinates for axisymmetric torus elements, with r being the radial and z being the longitudinal coordinate
  3) the x,y,z coordinates for three-dimensional tetrahedron and hexahedron elements

- Elements and nodes in the discretized FE model need to be identified by “element numbers” and “nodal numbers.” Element and node number are assigned in chronicle orders

- Elements are identified by the node numbers associated with the elements, e.g., Element 9 is associated with nodes 10, 11, 18, 17

- Nodes are identified by the assigned node numbers and their corresponding coordinates.

Example on FE mesh for a tapered bar:

- Total NO. of element: 18
- Total No. of nodes: 25
- Sliding boundary nodes: 8, 15, 22
- Fixed node: 1 (Minimum one)
- Nodes subject to applied forces: 7, 14, 21
- Nodes 1 to 7, 8 to 14 with equal space in x-coordinate, e.g., Node 1 at (0,0) Node 7 at (L/6,0). Node 8 at (0,H'), Node 14 at (L/6, H')There is no need to specify coordinates of the intermediate nodes
Discretization of solids for FE analysis

Discretization of highly complicated solid geometry are handled by sophisticated CAD packages. Solids with complicated geometry such as the 3-dimensional piston/connecting rod assembly and some others as presented below may involve combination of element of different sizes and shapes.
Step 2 Identify primary unknown quantity

**Primary unknown quantity** = The first and principal unknown quantity to be obtained by the FEM

- Frequently used **Primary unknown** quantities in FE analysis in mechanical engineering:
  - Stress analysis: Displacement \( \{u\} \) at notes
  - Heat transfer analysis: Temperature \( \{T\} \) at the nodes
  - Fluid dynamics analysis: Velocity \( \{v\} \) at nodes

- Other related unknown quantities, known as the “**secondary unknowns**” may be obtained from the primary unknown quantity.

**For example:**
In stress analysis, the primary unknowns are **nodal displacements**, but secondary unknown quantities include: **strains in elements** can be obtained by the “strain-displacement relations,” and the unknown **stresses in the elements** by the stress-strain relations (the Hooke’s law).
Interpolation functions and the derivation of Interpolation functions
- a very important step (Interpolation function is called “shape function in some literatures)

It is a very important part of the FE formulation because the quantities in the elements (e.g., the stresses) are the solutions that we are seeking, but these elements are interconnected at their nodes to approximate the original geometry by the discretized FE models. It is very important to derive a function that can relate the element quantities with the same quantities at the corresponding nodes.

Interpolation function in general cases:

Interpolation function in FEM relates the Element quantity $\Phi(x,y,z)$ and the 4 corresponding Nodal quantities: $\Phi_1, \Phi_2, \Phi_3, \Phi_4$ for tetrahedron elements for 3-D solids
We have learned there are basically 4 different types of elements on which the FE formulations are derived. These are:

1) Tetrahedron elements for 3-D solids with 4 nodes
2) Triangular plate elements for 2-D plane solids with 3 nodes
3) Bar elements for 1-D solids with 2 nodes
4) Axisymmetric triangular elements for solids with circular geometry with 3 nodes

Other element shapes: hexahedron with 6 sides 8 nodes, Quadrilateral plate elements with 4 sides, 4 nodes, and Axisymmetric quadrilateral cross-section elements with 4 sides, 4 nodes can be treated as assembly of the aforementioned basic element shapes.

We will deal with Types 1), 2) and 3) elements here. Interpolation functions relate the primary quantities of the element and the nodes for these 3 types of elements are shown below:

**Element Quantity** $\phi(x, y, z) = \text{Interpolation Function} \ N(x, y, z) \times \text{Nodal Quantity}$, $\{\phi\}^T = \begin{bmatrix} \phi_1 & \phi_2 & \phi_3 & \phi_4 \end{bmatrix}$ for 3-D tetrahedron elements with 4 nodes

**Element Quantity** $\phi(x, y) = \text{Interpolation Function} \ N(x, y) \times \text{Nodal Quantity}$, $\{\phi\}^T = \begin{bmatrix} \phi_1 & \phi_2 & \phi_3 \end{bmatrix}$ for 2-D plane elements with 3 nodes

**Element Quantity** $\phi(x) = \text{Interpolation function} \ N(x) \times \text{Nodal Quantity}$, $\{\phi\}^T = \begin{bmatrix} \phi_1 & \phi_2 \end{bmatrix}$ for 1-D bar elements with 2 nodes
There are different forms of interpolation functions used in FEM. We will present the most simple form for this class – the linear interpolation function. The elements using the linear interpolation functions are called “Simplex elements.”

The interpolation function of the Simplex elements are simplest form to use, and it is the most commonly used in FE formulation.

The size and shape of interpolation functions vary depending on the nature and the primary unknown quantities used in the FE analysis.

For example, take the simple triangular plate element as shown below:

The primary unknown quantity is the temperature $T$ with $\Phi(x,y) = T(x,y)$, the corresponding nodal quantities are: $\Phi_1 = T_1$, $\Phi_2 = T_2$, and $\Phi_3 = T_3$.

Because there is only ONE nodal value, i.e., the temperature involved, we may assume the following linear polynomial functions to relate the element temperature $T(x,y)$ and the corresponding nodal values as follows:

$$T(x,y) = \alpha_1 + \alpha_2 x + \alpha_3 y$$

Leading to the nodal values:

- $T_1 = \alpha_1 + \alpha_2 x_1 + \alpha_3 y_1$ for Node 1
- $T_2 = \alpha_1 + \alpha_2 x_2 + \alpha_3 y_2$ for Node 2
- $T_3 = \alpha_1 + \alpha_2 x_3 + \alpha_3 y_3$ for Node 3

where $\alpha_1$, $\alpha_2$, and $\alpha_3$ are the three constant coefficients.
Step 3 – Cont’d

Interpolation functions for simplex elements

The primary unknown quantity in this case is nodal displacements with the element DISPLACEMENTS designated by \( U(x,y) \) on the \((x-y)\) plane, or \( \Phi(x,y) = U(x,y) \)

But we realize fact that there are two components of displacements for the elements – one along the x-axis, \( U_x(x,y) \) and the other along the y-axis, \( V_y(x,y) \)

The same applies to the corresponding displacements at nodes with: one along the x-direction and the other along the y-direction.

Because there are two components of the unknown displacements in the element, we need to assume two linear polynomial functions to describe their variations within the element. The simplest form of functions are linear polynomials as shown below:

\[
\begin{align*}
U(x,y) &= \alpha_1 + \alpha_2x + \alpha_3y \\
V(x,y) &= \alpha_4 + \alpha_5x + \alpha_6y
\end{align*}
\]

where \( \alpha_1, \alpha_2, \ldots, \alpha_5, \alpha_6 \) are the six constant coefficients that can be related to the given coordinates of 3 nodes as follows:

If we let the nodal displacements to be represented by \( \{u\} \) with: \( u_1(x_1,y_1) = \Phi_1(x_1,y_1) \), for Node 1, etc. then we will have:

\[
\{u_1\} = \begin{bmatrix} u_{1x}(x_1,y_1) \\ u_{1y}(x_1,y_1) \end{bmatrix} = \begin{bmatrix} \alpha_1 + \alpha_2x_1 + \alpha_3y_1 \\ \alpha_4 + \alpha_5x_1 + \alpha_6y_1 \end{bmatrix} \quad \text{for Node 1, } \quad \{u_2\} = \begin{bmatrix} u_{2x}(x_2,y_2) \\ u_{2y}(x_2,y_2) \end{bmatrix} = \begin{bmatrix} \alpha_1 + \alpha_2x_2 + \alpha_3y_2 \\ \alpha_4 + \alpha_5x_2 + \alpha_6y_2 \end{bmatrix} \quad \text{for Node 2, } \quad \{u_3\} = \begin{bmatrix} u_{3x}(x_3,y_3) \\ u_{3y}(x_3,y_3) \end{bmatrix} = \begin{bmatrix} \alpha_1 + \alpha_2x_3 + \alpha_3y_3 \\ \alpha_4 + \alpha_5x_3 + \alpha_6y_3 \end{bmatrix} \quad \text{for Node 3}
\]

in which \( u = \text{displacement component along the } x\)-coordinate and \( v = \text{displacement component along the } y\)-coordinate
Step 3 – Cont’d

**Interpolation functions for 3-D tetrahedron elements**

For 3-D general solids with tetrahedron elements relating the element quantity $\Phi(x,y,z)$ with the four (4) associate nodal quantities by linear polynomial functions with the form:

The primary quantity of the Element has three components as:

$$\{\phi(x, y, z)\}^T = \{\phi_x(x, y, z), \phi_y(x, y, z), \phi_z(x, y, z)\}$$

where $\Phi_x(x,y,z) = \text{component along the x-direction}$,
$\Phi_y(x,y,z) = \text{component along the y-direction}$,
$\Phi_z(x,y,z) = \text{component along the z-direction}$

We further assume that the each of the three components of the element primary quantity is represented by LINEAR POLYNOMIAL functions as:

$$\phi_x(x, y, z) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z$$

$$\phi_y(x, y, z) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z$$

$$\phi_z(x, y, z) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z$$

in which $\alpha_1, \alpha_2, \alpha_3,$ and $\alpha_4$ are constants to be determined by the GIVEN nodal coordinates in corresponding FE mesh of the discretized solid. For instance: $(x_1,y_1,z_1)$ for Node $\Phi_1$; $(x_2,y_2,z_2)$ for Node $\Phi_2$; $(x_3,y_3,z_3)$ for Node $\Phi_3$; and $(x_4,y_4,z_4)$ for Node $\Phi_4$.

The total number of primary unknown quantities in a tetrahedron element with 4 nodes is 12, with each node consisting of 3 unknown quantities along the x-, y- and z-coordinate, e.g., $\Phi_{1x}, \Phi_{1y}, \Phi_{1z}$ for Node $\Phi_1$ along the respective x-, y- and z-coordinate.
Interpolation functions for 3-D tetrahedron elements

We may thus express the matrix of primary unknown of a tetrahedron element in terms of those at the associate 4 nodes as:

\[
\begin{bmatrix}
\phi_{1x} \\
\phi_{1y} \\
\phi_{1z} \\
\phi_{2x} \\
\phi_{2y} \\
\phi_{2z} \\
\phi_{3x} \\
\phi_{3y} \\
\phi_{3z} \\
\phi_{4x} \\
\phi_{4y} \\
\phi_{4z}
\end{bmatrix}
= \begin{bmatrix}
\phi_1(x, y, z) \\
\phi_2(x, y, z) \\
\phi_3(x, y, z) \\
\phi_4(x, y, z)
\end{bmatrix}
= \begin{bmatrix}
N_1(x, y, z) \\
N_2(x, y, z) \\
N_3(x, y, z) \\
N_4(x, y, z)
\end{bmatrix}
\]

The nodal quantity matrix has the form: \( \{\phi\}^T = \{\phi_1 \ \phi_y \ \phi_z \ \phi_2 \ \phi_y \ \phi_z \ \phi_3 \ \phi_y \ \phi_z \ \phi_4 \ \phi_y \ \phi_z \} \) are the primary quantities at Nodes \( \Phi_1, \Phi_2, \Phi_3, \) and \( \Phi_4 \), as indicated in the first subscript, with their second subscript indicating the directions in x-, y-, and z-directions.

The elements \( N_1, N_2, N_3 \) and \( N_4 \) in the above interpolation matrix denote the sub-matrices relating to Nodes \( \Phi_1, \Phi_2, \Phi_3 \) and \( \Phi_4 \) respectively.

It is convenient to express the interpolation function in the following form with one element associated with each node:

A row-matrix:

\[
\{N(x, y, z)\} = \begin{bmatrix}
N_1(x, y, z) \\
N_2(x, y, z) \\
N_3(x, y, z) \\
N_4(x, y, z)
\end{bmatrix}
\]

(1.5)
We define **Interpolation function** in FEM relates the **Element quantity** $\Phi(x,y,z)$ and the corresponding **Nodal quantities**: $\Phi_1$, $\Phi_2$, and $\Phi_3$.

Take for example of a triangular plate element:

![Diagram of a triangular plate element]

Primary unknown $\Phi(x,y)$ in ELEMENT:

Primary unknowns $\Phi_1$, $\Phi_2$, and $\Phi_3$ at three corresponding NODES $x_1$, $x_2$, $x_3$ and $y_1$, $y_2$, $y_3$ are “fixed coordinates.”

We assume a “linear function” for relating $\Phi(x,y)$ and $\Phi_1$, $\Phi_2$ and $\Phi_3$:

$$
\phi(x, y) = \alpha_1 + \alpha_2 x + \alpha_3 y
$$

$$
= \begin{bmatrix} 1 & x & y \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{bmatrix} = \{R\}^T \{\alpha\}
$$

(10.1)*

with $\{R\}^T = \begin{bmatrix} 1 & x & y \end{bmatrix}$

where $\alpha_1$, $\alpha_2$, and $\alpha_3$ are constants.

*Equation number in the assigned textbook*
Example on Derivation of Interpolation function for a planar solid – plate element

Because the coordinates \((x_1, y_1), (x_2, y_2)\) and \((x_3, y_3)\) of the nodes in a FE mode are fixed. We may substitute these coordinates into Equation (10.1) and obtain the following expressions for the corresponding quantities at the three nodes:

\[
\phi_1 = \alpha_1 + \alpha_2 x_1 + \alpha_3 y_1 \quad \text{for Node 1}
\]

\[
\phi_2 = \alpha_1 + \alpha_2 x_2 + \alpha_3 y_2 \quad \text{for Node 2}
\]

\[
\phi_3 = \alpha_1 + \alpha_2 x_3 + \alpha_3 y_3 \quad \text{for Node 3}
\]

or in a matrix form:

\[
\{\phi\} = [A]\{\alpha\}
\]

and

\[
\{\alpha\} = [A]^{-1}\{\phi\} = [h]\{\phi\}
\]

The matrix \([A]\) in Equations (1.11) and (1.12) contains the coordinates of the three nodes as:

\[
[A] = \begin{bmatrix}
1 & x_1 & y_1 \\
1 & x_2 & y_2 \\
1 & x_3 & y_3
\end{bmatrix}
\]
The inversion of matrix \([A]^{-1} = [h]\) can be performed to give:

\[
[h] = \frac{1}{|A|} \begin{bmatrix}
  x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\
  y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\
  x_3 - x_2 & x_1 - x_3 & x_2 - x_1
\end{bmatrix}
\]  

(1.13)

where \(|A|\) is the determinant of the element of matrix \([A]\)

\[
= (x_1y_2 - x_2y_1) + (x_2y_3 - x_3y_2) + (x_3y_1 - x_1y_3) = \text{the area of the element made of triangle } (\phi_1\phi_2\phi_3)
\]
By substituting (1.13) into (1.12) and then (1.10), the element quantity represented by \( \Phi(x,y) \) can be made to equal

\[
\phi(x, y) = \{ R \}^T \{ h \} \{ \phi \}
\]

We will thus have the interpolation function: \( N(x,y) = \{ R \}^T \{ h \} \) with \( \{ R \}^T = \{ 1 \ x \ y \} \) and \( \{ h \} \) given in Equation (1.13).

We thus have the relationship between the element quantity to the nodal quantifies by the following expression:

\[
\Phi(x,y) = \{ N(x,y) \} \{ \Phi \}
\]

or express the above equation in the form according to Equation (1.5) as:

\[
\begin{align*}
\{ \phi(x, y) \} = & \{ N_1(x, y) \} \{ \phi_1 \} + \{ N_2(x, y) \} \{ \phi_2 \} + \{ N_3(x, y) \} \{ \phi_3 \} \\
& \text{Interpolation function}
\end{align*}
\]

where \( N_1(x,y), N_2(x,y) \) and \( N_3(x,y) \) are the elements of interpolation functions for the corresponding nodes \( \Phi_1, \Phi_2, \text{and} \Phi_3 \).

The element displacements can thus be expressed in terms of the corresponding nodal displacements as:

\[
\{ \phi(x, y) \} = \{ N_1(x, y) \} \{ \phi_1 \} + \{ N_2(x, y) \} \{ \phi_2 \} + \{ N_3(x, y) \} \{ \phi_3 \}
\]

in which \( \{ \phi_1 \} = \{ \phi_{1x}(x_1, y_1), \phi_{1y}(x_1, y_1) \} \), \( \{ \phi_2 \} = \{ \phi_{2x}(x_2, y_2), \phi_{2y}(x_2, y_2) \} \), and \( \{ \phi_3 \} = \{ \phi_{3x}(x_3, y_3), \phi_{3y}(x_3, y_3) \} \) are the primary unknown quantities at the 3 nodes.
The 3 elements of the interpolation function \( N_1(x,y) \), \( N_2(x,y) \) and \( N_3(x,y) \) in Equation (3.1) have the forms (Ref.: “Applied FE analysis”, L.J. Segerlind, John Wiley & Sons, 1976):

\[
N_1(x,y) = \frac{1}{A} \left[ (x_2y_3 - x_3y_2) + (y_2 - y_3)x + (x_3 - x_2)y \right]
\]

\[
N_2(x,y) = \frac{1}{A} \left[ (x_3y_1 - x_1y_3) + (y_3 - y_1)x + (x_1 - x_3)y \right]
\]

\[
N_3(x,y) = \frac{1}{A} \left[ (x_1y_2 - x_2y_1) + (y_1 - y_2)x + (x_2 - x_1)y \right]
\]

(3.2)

where 
\[
A = (x_1y_2 - x_2y_1) + (x_2y_3 - x_3y_2) + (x_3y_1 - x_1y_3) = \text{the area of the element made of triangle } (\phi_1\phi_2\phi_3)
\]

**Numerical example:** Find the interpolation function of the triangular plate with specified nodal coordinates:

\( x_1 = 2, \ x_2 = 6, \ x_3 = 5; \ \text{and} \ y_1 = 1, \ y_2 = 4, \ y_3 = 8 \)

We find the area of the triangle \( \Phi_1\Phi_2\Phi_3 \) to be: \( A = 19 \text{ sq. unit} \)

The elements of the interpolation function \( \{N_1(x,y) \ \ N_2(x,y) \ \ N_3(x,y)\} \) have the following values by using expressions in Equation (3.2):

\[
N_1(x,y) = 1.4737 - 0.21052x - 0.0526y; \quad N_2(x,y) = -0.5789 + 0.3684x - 0.1578y; \quad N_3(x,y) = 0.10526 - 0.15789x + 0.210526y
\]
**Solution:**

Because the primary quantity in the element varies according to the \(x,y\) coordinates within the element, so the interpolation function is \(N(x,y)\).

We will use Equation (1.15) to determine the interpolation function \([N(x,y)]\) for the triangular plate element in the figure, with the given nodal coordinates: \(x_1 = 2, x_2 = 6, x_3 = 5\), and \(y_1 = 1, y_2 = 4\) and \(y_3 = 8\).

\[
\phi_1 = \alpha_1 + \alpha_2 x_1 + \alpha_3 y_1 \\
\phi_2 = \alpha_1 + \alpha_2 x_2 + \alpha_3 y_2 \\
\phi_3 = \alpha_1 + \alpha_2 x_3 + \alpha_3 y_3
\]

Leads to

\[
[A] = \begin{bmatrix}
1 & x_1 & y_1 \\
1 & x_2 & y_2 \\
1 & x_3 & y_3
\end{bmatrix} = \begin{bmatrix}
1 & 2 & 1 \\
1 & 6 & 4 \\
1 & 5 & 8
\end{bmatrix}
\] (a)
**Step 3-cont’d**  Numerical Example on expressing the interpolation function of a simplex triangular planar element

Compute the determinant:

$$|A| = \begin{vmatrix} 1 & 2 & 1 \\ 1 & 6 & 4 \\ 1 & 5 & 8 \end{vmatrix} = 19$$

Compute the \([h]\) matrix using Equation (1.13) with:

$$[h] = \frac{1}{19} \begin{bmatrix} 28 & -11 & 2 \\ -4 & 7 & -3 \\ -1 & -3 & 4 \end{bmatrix}$$

With the \([R]T = \{1 \ x \ y\}\) in Equation (1.10) and the \([h]\) shown above, we may compute the interpolation function for the triangular plate element from Equation (1.15) to be:

$$[N(x, y)] = [R]^T[h] = \begin{bmatrix} 1 & x & y \end{bmatrix} \begin{bmatrix} 28 & -11 & 2 \\ -4 & 7 & -3 \\ -1 & -3 & 4 \end{bmatrix} = \{1.4737 - 0.2105x - 0.0526y, -0.5789 + 0.3684x - 0.1579y, 0.1053 - 0.1579x + 0.2105y\}$$

We realize that the above results turn out to be identical with those computed from Equation (3.2) offered in the reference book by Segerlind.
Step 3—cont’d Example on Derivation of Interpolation function for one-dimensional bar elements

Derive the interpolation function for a bar element in Figure 1.6 with assigned coordinates for Node 1 at A with x=x₁ and the coordinate of Node 2 at B with x=x₂ for node 2.

Assume the longitudinal deformations Φ(x), i.e., the elongation or contraction of the bar element follows a linear polynomial function as shown in Figure 1.6.

Solution:

We begin our derivation with an assumption of using linear polynomial function for the interpolation function of the form:

\[ \phi(x) = \alpha_1 + \alpha_2 x \]  

where \( \alpha_1 \) and \( \alpha_2 \) are constants.

By comparing Equation (a) with Equation (1.10), we have the matrix:

\[ \{R\}^T = \begin{bmatrix} 1 & x \end{bmatrix} \]

Figure 1.6 Linear Interpolation Function of a Bar Element
We will need to derive the $[h]$ matrix in Equation (1.13) by the following computations:

We have: $\phi_1 = \alpha_1 + \alpha_2 x_1$ and $\phi_2 = \alpha_1 + \alpha_2 x_2$

From which we will have

$$\{\phi\} = \begin{bmatrix} \phi_1 \\ \phi_2 \end{bmatrix} = \begin{bmatrix} 1 & x_1 \\ 1 & x_2 \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \end{bmatrix} = [A] \{a\}$$

with the matrix $[A]$ to be:

$$[A] = \begin{bmatrix} 1 & x_1 \\ 1 & x_2 \end{bmatrix} \quad \text{and} \quad |A| = \begin{vmatrix} 1 & x_1 \\ 1 & x_2 \end{vmatrix} = x_2 - x_1$$

The inverse matrix of $[A]$ is:

$$[A]^{-1} = \frac{1}{x_2 - x_1} \begin{bmatrix} x_2 & -x_1 \\ -1 & 1 \end{bmatrix} = [h]$$

By following Equation (1.15), we have the interpolation function of a bar element to be:

$$N(x) = [R]^T [h] = \{1 \quad x\} \left( \frac{1}{x_2 - x_1} \begin{bmatrix} x_2 & -x_1 \\ -1 & 1 \end{bmatrix} \right) = \frac{1}{L} \{(x_2 - x) \quad (x_1 + x)\}$$

For the present case with $x_1 = 0$, and $x_2 = L$, we have the interpolation function to be:

$$N(x) = \begin{bmatrix} 1 - \frac{x}{L} \\ \frac{x}{L} \end{bmatrix}$$  \hspace{1cm} (1.7)
The interpolation function in Equation (1.7) may be expressed in the form shown in Equation (1.5) as:

\[ \phi(x) = \alpha_1 + \alpha_2 x \]

\[ \{N(x)\} = \{N_1(x) \quad N_2(x)\} \quad \text{with} \quad N_1(x) = 1 - \frac{x}{L}, \quad \text{and} \quad N_2(x) = \frac{x}{L} \]

We thus can express the element displacement in terms of the displacement of its associated nodes in the following expression:

\[ \phi(x) = \{N_1(x) \quad N_2(x)\} \begin{bmatrix} \phi_A \\ \phi_B \end{bmatrix} \]

(1.8)

**Example on Derivation of Interpolation function for one-dimensional bar elements**

The interpolation function in Equation (1.7) may be expressed in the form shown in Equation (1.5) as:

\[ \phi(x) = \alpha_1 + \alpha_2 x \]

\[ \{N(x)\} = \{N_1(x) \quad N_2(x)\} \quad \text{with} \quad N_1(x) = 1 - \frac{x}{L}, \quad \text{and} \quad N_2(x) = \frac{x}{L} \]

We thus can express the element displacement in terms of the displacement of its associated nodes in the following expression:

\[ \phi(x) = \{N_1(x) \quad N_2(x)\} \begin{bmatrix} \phi_A \\ \phi_B \end{bmatrix} \]

(1.8)

**Numerical example**

A 2-meter long slender bar is hinged to a support at point A as shown in the figure below. A force \( F = 100 \) Newton is applied to the bar at the other end B. Determine the interpolation function of the bar element.

**Solution:**

We have a bar element that is 2 m long. The bar element has two nodes located at A and B. If the primary quantity in the bar element is \( \Phi(x) \) with axis A along the longitudinal direction of the bar, and the same primary quantity at the associate two nodes to be: \( \Phi_A \) and \( \Phi_B \). The element quantity and the corresponding nodal values will be related by the interpolation function as shown in Equation (1.8), with: \( N_1(x) = 1 - x/2 \) and \( N_2(x) = x/2 \). Equation (1.8) for the current problem is:

\[ \phi(x) = \begin{bmatrix} 1 - \frac{x}{2} \\ \frac{x}{2} \end{bmatrix} \begin{bmatrix} \phi_A \\ \phi_B \end{bmatrix} \]
Step 4 Derivation of Element equation

The element equation relates the induced primary unknown quantity in the analysis with the action.

### Actions and Induced Reactions

<table>
<thead>
<tr>
<th>Analysis</th>
<th>Actions ( { P } )</th>
<th>Induced Reactions ( { \Phi } )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress analysis</td>
<td>Forces ( { F } )</td>
<td>Primary unknown: Displacement at nodes ( { u } )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Secondary unknowns: Stresses ( { \sigma } )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Strains ( { \varepsilon } )</td>
</tr>
<tr>
<td>Heat conduction analysis</td>
<td>Thermal forces ( { Q } )</td>
<td>Primary unknown: Temperature at nodes</td>
</tr>
<tr>
<td>Fluid mechanics analysis</td>
<td>Pressure or head ( { P } )</td>
<td>Primary unknown: Velocity at nodes</td>
</tr>
</tbody>
</table>

There are generally two methods to derive the element equations:

1) The Rayleigh-Ritz method, and

2) The Galerkin method
Step 4 –Cont’d  The Rayleigh-Ritz method

This method is used for problems that cannot be described by single differential equation with appropriate boundary conditions. Such as stress analysis of solid structures.

It requires the derivation of functional $\chi(\Phi)$ for the particular analysis, and minimize this functional with respect to the induced reaction of the system by using the Variational process as described in Chapter 2.

In FE Model with finite number of ELEMENTS interconnected at NODES:

Actions $\{P\}$: $p_1$, $p_2$, $p_3$,...,$P_n$ in discretized continuum

Induced Reactions: $\{\phi\}^e$: $\phi_1^e$, $\phi_2^e$, $\phi_3^e$,...,$\phi_m^e$ in ELEMENTS of discretized continuum

The functional of the discretized continuum is:

$$\chi(\phi) = \sum_{m=1}^{m} \chi_m^e(\phi)$$

where $\chi_m^e(\phi)$ is the functional of Element $m$

The Variational process required for the equilibrium of the discretized continuum becomes:

$$\frac{\partial \chi(\phi)}{\partial \{\phi\}} = \sum_{1}^{m} \frac{\partial \chi_m^e(\phi)}{\partial \{\phi\}^e} = 0$$

From which the “element equation” for every element ($m = 1, 2, 3,....$) in the discretized FE model is derived
The Rayleigh-Ritz method

The Element equations derived form the expressions:

\[ \sum_{i}^{m} \frac{\partial \chi_{m}^{e}(\phi_{m})}{\partial \{\phi_{m}^{e}\}} = 0 \]

with

\[ \frac{\partial \chi_{1}^{e}(\phi_{1}^{e})}{\partial \{\phi_{1}^{e}\}} = 0, \quad \frac{\partial \chi_{2}^{e}(\phi_{2}^{e})}{\partial \{\phi_{2}^{e}\}} = 0, \quad \ldots, \quad \frac{\partial \chi_{m}^{e}(\phi_{m}^{e})}{\partial \{\phi_{m}^{e}\}} = 0 \]

for each element in the discretized FE mesh will have a general form called “Element Equation,” as:

\[ [K_{e}]{q} = \{Q\} \]

(1.27)

where

- \([K_{e}] = \text{Element matrix}\)
- \({q} = \text{Vector of primary unknown quantities at the nodes of the element}\)
- \({Q} = \text{Vector of element nodal actions (e.g., forces)}\)
Step 4 – Cont’d

Galerkin method

In contrast to the Rayleigh-Ritz method, this method is used to derive the element equations for the cases in which specific differential equations with appropriate mathematical expressions for the boundary conditions available for the analytical problems, such as heat conduction and fluid dynamic analyses.

Real Situation on solids

Differential Equation: \( D(\Phi) \) for the volume \( V \)

Boundary condition: \( B(\Phi) \) for the real situation on boundary \( S \)

Mathematical model: \( \int V W D(\phi) dV + \int S W B(\phi) ds = 0 \)

where \( W \) and \( \bar{W} \) are arbitraly weighting functions

Galerkin method lets \( W_j \) and \( \bar{W}_j = N(r) \) and let \( R \) to be minimum, or \( R \to 0 \) for good discretization, resulting in:

\[ [K_e] \{q\} = \{Q\} \]

Approximate situation: Discretized Situation with elements

Element \( \Phi(r) \)

Nodal \( \{\Phi\} \)

\( \Phi(r) = N(r)\Phi \)

Differential Equation: \( D(N(r)\Phi) \) for the element volume \( V \)

Boundary condition: \( B(N(r)\Phi) \) for the real situation on element boundary

Mathematical model: \( \int V W_j D(\sum N_i(r)\phi_i) dV + \int S \bar{W}_j B(\sum N_i(r)\phi_i) ds = R \)

\( W_j \) and \( \bar{W}_j \) are discretized weighting functions, and \( R \) is the residual
Step 5 Derive overall Stiffness Equation

This step assembles all individual element equations derived in Step 4 to provide the “Stiffness equations” for the entire medium.

Mathematically, this equation has the form:

\[
[K]\{q\} = \{R\}
\]

(1.28)

where \( [K] \) = overall stiffness matrix = \( \sum_{e=1}^{M} [K_e] \)

\( M \) = total number of elements in the discretized FE mesh

\( \{R\} \) = assemblage of resultant vector of nodal actions

Caution: In assembly \( [K] \) from \( [k_e] \) for those nodes that are shared by more than one elements, such as Nodes 9-13 and 15-18 in the FE mesh for a tapered bar in the example in Step 1. The nodal entries for these nodes should be added algebraically, as will be demonstrated in the following case [Hsu and Sinha – the reference book on page 224-226]
A 4-side plate structure is sub-divided by 4 triangular elements as shown in the figure below:

The plate is allowed to deform in both x- and y-directions, so each node will be associated with two displacement components too, i.e., \( \{u\} = u_x \) and \( u_y \).

So, all together, we should have total nodal unknown quantities of \( 5 \times 2 = 10 \), which is also the size of the \([K]\) matrix (10x10).

Figure to the right:
For element 1: Row 1, 2 for values of \( u_{1x}, u_{1y} \) for Node 1
Row 3, 4 for values of \( u_{2x}, u_{2y} \) for Node 2
Row 7, 8 for values of \( u_{4x}, u_{4y} \) for Node4

Follow the same pattern for other elements. Add all values to fill all 10x10 entries in the assemble \([K]\) and \([R]\) matrices.
**Step 6 Solve for primary unknowns**

Use the inverse matrix method to solve the primary unknown quantities \( \{q\} \) at all the nodes from the overall stiffness equations

\[
\{q\} = [K^{-1}]\{R\} \quad (1.29)
\]

Or use the Gaussian elimination method or its derivatives to solve nodal quantities \( \{q\} \) from the equation:

\[
[K]\{q\} = \{R\} \quad (1.28)
\]

by converting the \([K]\) matrix to upper triangular matrix with appropriate modification of the \(\{R\}\) matrix.

In FEA, additional interchange of rows and columns of all the three matrices in Equation (1.28) take place due to the fact that some nodes in the FE model either have zero or specified values (See nodes 1, 8, 15, 22 in the tapered bar in Step 1, with node 1 having both displacement components to be zero, and the other three nodes having their u\(_x\) = re-arrangement of the three 0). In such cases, these unknowns in the \(\{q\}\) matrix do not require computations to determine their values.

The following re-arrangement of the three matrices in Equation (1.28) are made for solving the primary unknowns in Equation (1.28):

\[
\begin{bmatrix}
  K_{aa} & K_{ab} \\
  K_{ba} & K_{bb}
\end{bmatrix}
\begin{bmatrix}
  q_a \\
  q_b
\end{bmatrix} =
\begin{bmatrix}
  R_a \\
  R_b
\end{bmatrix}
\]

where \(\{q_a\}\) = specified (known) nodal quantities; \(\{R_b\}\) = specified (known) applied resulting actions, from which we may obtain:

\[
\{q_b\} = [K_{bb}]^{-1} (\{R_b\} - [K_{ba}]\{q_a\}) \quad (6.12, \text{REF})
\]
Example: Use the matrix partition method to solve the following set of equations [Hsu and Sinha 1992]:

\[
\begin{bmatrix}
5 & 4 & -3 & 5 & -6 \\
4 & -5 & 4 & -4 & 3 \\
2 & 3 & -4 & 5 & -6 \\
-1 & -2 & 3 & -4 & 5 \\
-7 & 6 & -5 & 4 & -3
\end{bmatrix}
\begin{bmatrix}
u_1 \\
u_2 \\
u_3 \\
u_4 \\
u_5
\end{bmatrix}
=
\begin{bmatrix}
-15 \\
39 \\
-37 \\
33 \\
-55
\end{bmatrix}
\]

with \( u_3 = 7 \), \( u_4 = 2 \) and \( u_5 = 6 \)

By following the procedure presented above, the original matrix equation is partitioned into the following form:

\[
\begin{bmatrix}
-4 & 5 & 3 & -1 & -2 \\
4 & -3 & -5 & -7 & 6 \\
5 & -6 & -4 & 2 & 3 \\
5 & -6 & -3 & 4 & 4 \\
-4 & 3 & 4 & 4 & -5
\end{bmatrix}
\begin{bmatrix}
u_1 \\
u_2
\end{bmatrix}
=
\begin{bmatrix}
2 \\
6 \\
7 \\
-15 \\
39
\end{bmatrix}
\]

With the following sub-matrices:

\[
[k_{aa}] = \begin{bmatrix}
-4 & 5 & 3 \\
4 & -3 & -5 \\
5 & -6 & -4
\end{bmatrix},
[k_{ab}] = \begin{bmatrix}
-1 & -2 \\
-7 & 6 \\
2 & 3
\end{bmatrix},
[k_{ba}] = \begin{bmatrix}
5 & 4 \\
4 & -5
\end{bmatrix}
\]

\[
[q_b] = \begin{bmatrix}
u_1 \\
u_2
\end{bmatrix}
=[5, 4]^{-1}
\begin{bmatrix}
-15 \\
39
\end{bmatrix}
- \begin{bmatrix}
4 & -6 & -3 \\
-4 & 4 & 4
\end{bmatrix}
\begin{bmatrix}
2 \\
6
\end{bmatrix}
\]

\[
=[0.12195, 0.09756]
\begin{bmatrix}
32 \\
1
\end{bmatrix}
= \begin{bmatrix}
3.99996 \\
2.99997
\end{bmatrix}
\]

Solve for the 2 unknowns: \( u_1 \) and \( u_2 \).
Step 7 Solve for secondary unknowns

For secondary unknowns in stress analysis by FEM:

**The primary unknowns** = \{q\} = \{u\} = displacement components at all nodes in the FE model

**Secondary unknowns:**

- **Strains in element** \{\varepsilon\} from the \{strain\} - \{displacement\} relations (derived from “Theory of Elasticity”)
- **Stresses in element** \{\sigma\} from the \{stress\} - \{strain\} relation: – the Hooke’s law for simple uni-axial stress case
  Generalized Hooke’s law by Theory of Elasticity
Step 8 Display and Interpretation of Results

Form 1: Tabulation of results
Form 2: Graphic displays: (1) Static with contours. (2) Animations

The regions in red indicate the induced von-Mises stresses exceed the allowable Strength of material → vulnerable for failure

Induced Stress Contours in a Gear Tooth by FE Analysis verified by experiments [Hsu and Sinha 1992]

Three Frames of an Animated Deflection of a Beam by FE Analysis [Hsu and Sinha 1992]
Summary on Steps in FEA

There are generally eight (8) steps in a FE Analysis:

**Step 1: Discretization of real continuum or structure** – (Establish the FE mesh)

- Use bar elements for 1-D, Triangular and quadrilateral elements for 2-D plane and axisymmetric structures, and tetrahedron and hexahedron elements for 3-D media.
- Assign element and nodal numbers
- Enter nodal coordinates
- Enter element descriptions, e.g. Nodes 9,1017, 16 for Element 8; Node 16, 17, 24,24 for Element 16, etc.
- Enter nodal actions and constraints
- Make sure to place many more but smaller elements in regions with drastic change of geometry
- Many commercial FE codes offer “automatic mesh generation.” Make use for this option.
- Many commercial packages offer automatic transformation of media profiles by CAD to FE analysis with automatic mesh generations. Again make use of this option.
Step 2: Identify primary unknown quantity:

- Element displacements for stress analysis
- Element temperature for heat conduction analysis
- Element velocities for fluid dynamic analysis

Step 3: Interpolation functions and derivation of Interpolation functions
- a very important step

Because primary unknown quantities in FEA are for those in the elements, but elements are interconnected at nodes, so it is important to establish relationship for the quantities in the elements with the associated nodes. This is what interpolation functions are defined. Mathematical expressions of interpolation functions:

\[
\{\phi(r)\} = \{N(r)\}\{\Phi\}
\]

where \(\{\Phi(r)\}\) = Element quantity, \(\{\Phi\}\) = nodal quantity, \(N(r)\) = interpolation function with \(r\) = coordinates

Interpolation functions may be expressed to relate the corresponding nodes in the following way:

- **Element Quantity** \(\phi(x, y, z)\) = Interpolation Function \(\{N_1(x, y, z)\ N_2(x, y, z)\ N_3(x, y, z)\ N_4(x, y, z)\}\) for tetrahedron elements with 4 nodes
  \[
  \{\phi\}^T = \{\phi_1 \ \phi_2 \ \phi_3 \ \phi_4\}
  \]

- **Element Quantity** \(\phi(x, y)\) = Interpolation Function \(\{N_1(x, y)\ N_2(x, y)\ N_3(x, y)\}\) for plate elements with 3 nodes
  \[
  \{\phi\}^T = \{\phi_1 \ \phi_2 \ \phi_3\}
  \]

- **Element Quantity** \(\phi(x)\) = Interpolation function \(\{N_1(x)\ N_2(x)\}\) for bar elements with 2 nodes
  \[
  \{\phi\}^T = \{\phi_1 \ \phi_2\}
  \]
Step 4: Derivation of Element equation

There are generally two methods to derive the element equations:

1) **The Rayleigh-Ritz method** for stress analysis of solid structure using the POTENTIAL ENERGY in deformed solids as the functional to be minimized, and

2) **The Galerkin method** for heat conduction analysis of solids and fluid dynamic analysis with identifiable distinct differential equations and boundary conditions

Element equations of the form: 

\[
[K_e]\{q\} = \{Q\}
\]

where $$[K_e] = \text{Element matrix}$$ 
$$\{q\} = \text{Vector of primary unknown quantities at the nodes of the element}$$  
$$\{Q\} = \text{Vector of element nodal actions (e.g., forces)}$$
Step 5: Derive Overall Stiffness Equation

\[ [K]\{q\} = \{R\} \]  \hspace{1cm} (1.28)

where \([K] = \text{overall stiffness matrix} = \sum_{i=1}^{M} [K_e] \)

- \(M = \text{total number of elements in the discretized FE mesh}\)
- \(\{R\} = \text{assemblage of resultant vector of nodal actions}\)

\textbf{Caution}: In assembly \([K]\) from \([k_e]\) for those nodes that are shared by more than one elements, the nodal entries for these nodes should be added algebraically, as demonstrated in a previous numerical example.

Step 6: Solve for primary unknowns from Overall Stiffness Equation

Partition the \([K]\) matrix as necessary:

\[
\begin{bmatrix}
K_{aa} & K_{ab} \\
K_{ba} & K_{bb}
\end{bmatrix}
\begin{bmatrix}
q_a \\
q_b
\end{bmatrix} =
\begin{bmatrix}
R_a \\
R_b
\end{bmatrix}
\]

where \(\{q_a\} = \text{specified (known) nodal quantities}\); \(\{R_b\} = \text{specified (known) applied resulting actions}\), from which we may obtain

So that the unknown primary unknown quantities are solved by the following equations:

\[
\{q_b\} = [K_{bb}]^{-1} (\{R_b\} - [K_{ba}]{q_a})
\]  \hspace{1cm} (6.12, REF)
Step 7: Solve for secondary unknowns

The secondary unknown quantities in FE analysis in mechanical engineering can be obtained by the primary unknown quantities solved by the FE analysis. For instance:

In stress analysis of solid structures: the strains in elements can be related to its element displacements, and the stresses element strains by using the Hooke’s law.

Step 8: Display and Interpretation of Results

Results of FEA usually are presented in the following forms: (1) tabulations, (2) graphics: static and animation

Interpretation of results:

(1) von-Mises stress in the output of FEA of stress analysis of solid structure represents the stress in elements with multi-axial stresses induced by the applied load. This stress needs to be kept below the “yield stress” of the material to avoid plastic deformation, and kept below the allowable stress of the material in order to avoid structure failure.

(2) Nodal displacements will relate to the deformation of the solid structure, which should be kept below the allowable amount set by the design requirement.
End of Chapter 3