2.0 CONCEPTS IN THE FINITE ELEMENT ANALYSIS OF AN ELASTOSTATIC SOLID CONTINUUM

Finite element method has been established as a powerful and versatile numerical procedure for analysing a variety of engineering problems. In the research, finite element method is used for the elastostatic analysis of a solid continuum like arch dams, which are modeled in a three dimensional domain. This method models a continuum as an assemblage of small elements of simple geometry that is easier to analyse than the actual structure. Thus, a complex solution is approximated by a model that consists of piece-wise continuous simple solutions.

2.1 STEPS OF FINITE ELEMENT ANALYSIS

Major steps involved in the finite element analysis of a structure are:

1. Discretisation of continuum into many sub regions called finite elements. Formulation of element properties, loads, deformations associated with degrees of freedom, boundary conditions.

2. Assemblage of elements to obtain finite element model of the structure.

3. Application of known loads in terms of nodal forces and moments.

4. Application of displacement boundary conditions.

5. Solution of simultaneous algebraic equations to determine nodal degrees of freedom, which are the basic unknowns of the problem.
6. Calculation of strains and stresses and output the results by an interpretation program.

The entire domain is discretised into a large number of sub regions of different sizes, shapes and orientations called finite elements. These finite elements are assumed to be connected to each other only at a finite number of points called ‘nodes’. The field variables; in the present case, displacements, are approximated by piecewise continuous functions. Nodal values of the field variables are treated as the unknowns and are determined by making use of related variational principles. Once the nodal displacements are determined, the secondary variables such as stresses and strains are obtained by means of strain-displacement and stress-strain relations.

### 2.1.1 Concept of Interpolation

Interpolation means approximation of the value of a function between known values by operating on the known values with a formula different from the function itself. In the finite element context, the known values are the degrees of freedom to be found by solving algebraic equations.

The nodal displacements of each element are assumed by known means. The nodal displacement vector for the element is given by:

\[
\{u^e\} = \begin{bmatrix} u_1 & v_1 & w_1 & u_2 & v_2 & w_2 & \ldots & u_n & v_n & w_n \end{bmatrix}^T
\] (2.1)

where \(u_i, v_i, w_i\) are the nodal displacement degrees of freedom. Knowing the element displacement vector \(\{u^e\}\), the interior displacement field within the element is obtained by interpolation.[21]
\[ \begin{bmatrix} u(x, y, z) \\ v(x, y, z) \\ w(x, y, z) \end{bmatrix} = \sum N_i u_i = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & \cdots & N_n & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 & \cdots & 0 & N_n & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 & \cdots & 0 & 0 & N_n \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ w_1 \\ \vdots \\ u_n \\ v_n \\ w_n \end{bmatrix} \]  

(2.2)

\[ \{u\} = N_1 u_1 + N_2 u_2 + \cdots + N_n u_n = [N] \{u^e\} \]  

(2.3)

where \([N]\) is called the interpolation or shape function matrix.

### 2.1.2 Strain Displacement Relation

Once the displacement vector at any point is known, the strain vector is obtained by making use of the strain displacement relations. For three-dimensional elastostatic problems these relations are given by:

\[ \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yx} \\ \gamma_{xz} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \end{bmatrix} \begin{bmatrix} u \\ v \\ w \end{bmatrix} \]  

(2.4)

or
\{\varepsilon\} = [L]\{u\} \tag{2.5}

= [L][N]\{\varepsilon^e\}

= [B]\{\varepsilon^e\}

where \([B]\), which relates the interior element strain field to the element nodal displacement vector, is very important in the finite element analysis. It is called the strain-displacement matrix.

### 2.1.3 Stress Strain Relation

The stress vector \(\{\sigma\}\) and the strain vector \(\{\varepsilon\}\) for a three dimensional homogeneous linearly elastic isotropic continuum is related to each other by the constitutive matrix \([D]\).

\[
\{\sigma\}_{(6 \times 1)} = [D]_{(6 \times 6)} \{\varepsilon\}_{(6 \times 1)} \tag{2.6}
\]

The matrix \([D]\) is given by:

\[
[D] = \begin{bmatrix}
(1 - \mu) & \mu & 0 & 0 & 0 \\
(1 - \mu) & \mu & 0 & 0 & 0 \\
(1 - \mu) & 0 & 0 & 0 & 0 \\
\frac{1 - 2\mu}{2} & 0 & 0 & \frac{1 - 2\mu}{2} & 0 \\
\frac{1 - 2\mu}{2} & 0 & \frac{1 - 2\mu}{2} & \frac{1 - 2\mu}{2} & \frac{1 - 2\mu}{2}
\end{bmatrix}
\]

\(\text{symmetric}\)

where, \(\bar{E} = \frac{E}{(1 + \mu)(1 - 2\mu)}\)

In the above \(E\) and \(\mu\) are the Young's modulus and Poisson's ratio respectively.

Thus, the stress field in the element can be found.
## 2.1.4 Finite Element Equations

Basic finite element equation can be arrived at by the principle of virtual work.[21] The virtual displacement field within the element by imposing a virtual displacement field can be written as:

\[ \delta u = [N]\{\delta u^e\} \] (2.8)

The virtual strain corresponding to the above element nodal virtual displacement vector \( \{\delta u^e\} \) is given by:

\[ \{\delta e\} = [B]\{\delta u^e\} \] (2.9)

By equating the internal virtual work done to the external virtual work,

\[ \int_{V_e} \{\delta e\}^T \{\sigma\} dV_e = \int_{V_e} \{\delta u^e\}^T \{b\} dV_e + \int_{S_e} \{\delta u^e\}^T \{p\} dS_e \] (2.10)

where, \( V_e \) and \( S_e \) are the element volume and surface area respectively; \( \{b\} \) is the body force vector and \( \{p\} \) denotes the surface force acting over the periphery of the element.

Making use of Eq. (2.7), (2.8) and (2.9) in Eq. (2.10) we get:

\[ \{\delta u^e\}^T \int_{V_e} B^T D B dV_e \{u^e\} = \{\delta u^e\}^T \left[ \int_{V_e} N^T \{b\} dV_e + \int_{S_e} N^T \{p\} dS_e \right] \] (2.11)

Since \( \{\delta u^e\} \) represents an arbitrary nodal virtual displacement vector, Eq. (2.11) can be written as:

\[ [k^e]\{u^e\} = \{r^e\} \] (2.12)

where

\[ [k^e] = \int_{V_e} B^T D B dV_e \] (2.13)

and

\[ \{r^e\} = \int_{V_e} N^T \{b\} dV_e + \int_{S_e} N^T \{p\} dS_e \] (2.14)

In the above, \([k^e]\) is called the element stiffness matrix and \(\{r^e\}\) is called the element load vector.
2.1.5 **Assembly of Element Matrices**

Element stiffness matrices and load matrices are combined by a process known as the *assembly*, to arrive at the finite element equations of the entire structure. Thus

\[
[K]\{U\} = \{R\} \quad \text{(2.15)}
\]

where,

\[
[K] = \sum_{e=1}^{n_{\text{Elems}}} [k^e] \quad \text{and}
\]

\[
\{R\} = \sum_{e=1}^{n_{\text{Elems}}} \{r^e\} \quad \text{(2.16)}
\]

Eq. (2.15) is a set of simultaneous linear algebraic equations. Once boundary conditions are prescribed, it can be solved to arrive at the global displacement vector \(\{U\}\) from which the element nodal displacement vector \(\{u^e\}\) can be extracted from which the displacement, strain and stress fields within the element can be found.

### 2.2 ISOPARAMETRIC FORMULATION

Isoparametric formulation is used to generate elements with irregular shapes and curved boundaries, which are needed in modelling arch dams.

#### 2.2.1 Concept of Isoparametric Formulation

The same set of polynomial interpolation functions are used for defining the geometry and interpolating the displacement field in an isoparametric element. Thus the components of the displacement vector at any interior point are interpolated from the nodal values as;
\[ u(\xi, \eta, \zeta) = \sum_{i=1}^{n} N_i u_i \]
\[ v(\xi, \eta, \zeta) = \sum_{i=1}^{n} N_i v_i \]
\[ w(\xi, \eta, \zeta) = \sum_{i=1}^{n} N_i w_i \]  
\hspace{1cm} (2.17)

where \( n \) is the number of nodes per element and \( N_i \) are the interpolation polynomials.

The isoparametric mapping is given by
\[ x(\xi, \eta, \zeta) = \sum_{i=1}^{n} N_i x_i \]
\[ y(\xi, \eta, \zeta) = \sum_{i=1}^{n} N_i y_i \]
\[ z(\xi, \eta, \zeta) = \sum_{i=1}^{n} N_i z_i \]  
\hspace{1cm} (2.18)

where, \( x_i, y_i, z_i \) are the global coordinates of the nodal points of the element.

### 2.2.2 Coordinate Transformation and Concept of Jacobian Matrix

Derivatives of the displacement field with respect to \( x, y, z \) coordinates are needed in order to determine the strain components. These are not readily available as the displacement field is known in terms of the local coordinates \( \xi, \eta, \zeta \) and not in terms of the global coordinates \( x, y, z \) as given by Eq. (2.17). Required derivatives are obtained by means of the following transformation which follows from the chain rule of partial differentiation.
\[
\begin{bmatrix}
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta} \\
\frac{\partial}{\partial \zeta}
\end{bmatrix}
\begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\
\frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta}
\end{bmatrix}
= [J] 
\frac{\partial}{\partial x} \\
\frac{\partial}{\partial y} \\
\frac{\partial}{\partial z}
\end{bmatrix}
\]

(2.19)

In the above \([J]\) is called the Jacobian matrix. It is obtained by making use of Eq. (2.18). Thus

\[
J = \left[ \sum_{i=1}^{N} \sum_{j=1}^{N} \sum_{k=1}^{N} N_{i,\xi} N_{j,\eta} N_{k,\zeta} \begin{bmatrix} x_1 & y_1 & z_1 \\ x_2 & y_2 & z_2 \\ \vdots & \vdots & \vdots \\ x_n & y_n & z_n \end{bmatrix} \right]
\]

= \begin{bmatrix} N_{1,\xi} & N_{2,\xi} & \ldots & N_{n,\xi} \\ N_{1,\eta} & N_{2,\eta} & \ldots & N_{n,\eta} \\ N_{1,\zeta} & N_{2,\zeta} & \ldots & N_{n,\zeta} \end{bmatrix} \begin{bmatrix} x_1 & y_1 & z_1 \\ x_2 & y_2 & z_2 \\ \vdots & \vdots & \vdots \\ x_n & y_n & z_n \end{bmatrix}
\]

(2.20)

### 2.2.3 Interpolation Functions

Standard eight, twenty, twenty seven noded and higher order hexahedral elements are proposed for modeling the three-dimensional elastic continuum. The interpolation functions used for the eight-noded elements are the Lagrangian tri-linear interpolation polynomials.[20,24,84] These are given by

\[
N_i = \frac{1}{8} (1 + \xi_i \eta_i \zeta_i) 
\]

where, \(\xi_i, \eta_i, \zeta_i\) are the local coordinates at node \(i\). The twenty-noded element belongs to the serendipity family and the interpolation polynomials are given by:

**Corner nodes:**

\[
N_i = \frac{1}{8} (1 + \xi_i \eta_i) (1 + \zeta_i \eta_i) (1 + \xi_i \zeta_i) \eta_i \zeta_i - 2
\]

\(i = 1 \text{ to } 8\)
Mid side nodes in $\xi$ direction

$$N_i = \frac{1}{4}(1-\xi^2)(1+\eta\eta_i)(1+\zeta\zeta_i) \quad i = 9, 11, 13, 15$$

Mid side nodes in $\eta$ direction

$$N_i = \frac{1}{4}(1-\eta^2)(1+\xi\xi_i)(1+\zeta\zeta_i) \quad i = 10, 12, 14, 16$$

Mid side nodes in $\zeta$ direction

$$N_i = \frac{1}{4}(1-\zeta^2)(1+\xi\xi_i)(1+\eta\eta_i) \quad i = 17, 18, 19, 20$$

(2.22)

The twenty seven noded elements belong to the Lagrangian tri-quadratic family and the interpolation polynomials are arrived.[24,25]

**Corner Nodes**

$$N_i = \frac{1}{8}(1+\xi\xi_i)(1+\eta\eta_i)(1+\zeta\zeta_i)(\xi\xi_i + \eta\eta_i + \zeta\zeta_i - 2)(\xi\xi_i \eta\eta_i \zeta\zeta_i)$$

Mid side nodes in $\xi$ direction

$$N_i = \frac{1}{4}(1-\xi^2)(1+\eta\eta_i)(1+\zeta\zeta_i) \eta\eta_i \zeta\zeta_i$$

Mid side nodes in $\eta$ direction

$$N_i = \frac{1}{4}(1-\eta^2)(1+\xi\xi_i)(1+\zeta\zeta_i) \xi\xi_i \zeta\zeta_i$$
Mid side nodes in $\zeta$ direction

$$N_i = \frac{1}{4}(1 - \zeta^2)(1 + \zeta\tilde{\xi},)(1 + \eta\tilde{\eta},) \eta\eta_i$$

Mid face nodes in $\xi$ direction

$$N_i = \frac{1}{2}(1 - \xi^2)(1 - \eta^2)(1 + \zeta\tilde{\eta},) \zeta\zeta_i$$

Mid face nodes in $\eta$ direction

$$N_i = \frac{1}{2}(1 - \xi^2)(1 - \eta^2)(1 + \zeta\tilde{\xi},) \zeta\zeta_i$$

Mid face nodes in $\zeta$ direction

$$N_i = \frac{1}{2}(1 - \xi^2)(1 - \eta^2)(1 + \zeta\tilde{\xi},) \zeta\zeta_i$$

Mid body node

$$N_i = (1 - \xi^2)(1 - \eta^2)(1 - \zeta^2$$

where $i = 1, 2, 3, \ldots, 27$ \hspace{1cm} (2.23)

### 2.2.4 Element Stiffness Matrix

Strains are obtained from displacements as

$$\{\varepsilon\} = [L] \{u\} \hspace{1cm} (2.24)$$

where, $\{u\} = [N] \{u^e\}$

Substituting in Eq.2.24

$$\{\varepsilon\} = [L] [N] \{u^e\} \hspace{1cm} (2.25)$$

$$\{\varepsilon\} = [B] \{u^e\}$$

Thus,

$$[B] = [L] [N] \hspace{1cm} (2.26)$$

$[B]$ is called strain-displacement transformation matrix.
where \( [L] = \begin{bmatrix}
\frac{\partial}{\partial x} & 0 & 0 \\
0 & \frac{\partial}{\partial y} & 0 \\
0 & 0 & \frac{\partial}{\partial z} \\
\frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\
0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\
\frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x}
\end{bmatrix} \) \hspace{1cm} (2.27)

and \( [N] = \begin{bmatrix}
N_1 & 0 & 0 & N_2 & 0 & 0 & \cdots & N_n & 0 & 0 \\
0 & N_1 & 0 & 0 & N_2 & 0 & \cdots & 0 & N_n & 0 \\
0 & 0 & N_1 & 0 & 0 & N_2 & \cdots & 0 & 0 & N_n
\end{bmatrix} \) \hspace{1cm} (2.28)

\([N]\) expressed in terms of \(\xi, \eta, \zeta\) coordinates can be changed to \(x, y, z\) coordinates by invoking chain rule for \(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\). The element stiffness matrix \([k^e]\) from the principle of virtual work as arrived earlier is then:

\[
[k^e] = \int \int \int B^T DB \, dV_e = \int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} B^T DB \, dx \, dy \, dz = \int_{-1}^{1} \int_{-1}^{1} B^T DB \, J \, d\xi \, d\eta \, d\zeta \hspace{1cm} (2.29)
\]

The above integral is evaluated by using numerical integration method. The Gauss integration method is most widely used in finite element analysis, for evaluating the integral of a function.[21, 25]

\[
\int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} \Phi(\xi, \eta, \zeta) \, d\xi \, d\eta \, d\zeta = \sum_{i} \sum_{j} \sum_{k} W_i W_j W_k \Phi(\xi, \eta, \zeta) \hspace{1cm} (2.30)
\]

Using Gauss quadrature rule the above integral in Eq. (2.29) will become:

\[
[k^e] = \sum_{i} \sum_{j} \sum_{k} W_i W_j W_k B^T DB(\xi, \eta, \zeta) J \hspace{1cm} (2.31)
\]