The Fundamentals

What is Finite Element (FE) Method or Analysis?

A matrix solution technique initially developed by structural engineers at Stuttgart (Germany), Berkeley, and NASA. It was further developed by mathematical theories.

Trivia: The term "Finite Element" was coined by Ray W. Clough of Berkeley.

Modeling: Dividing a structure or a continuum into a series of discrete, well-formulated entities (known as Elements).

Analysis: Approximation of field quantities within these discrete elements and combining the elements into an approximate behavior of the initial physical problem.

Examples of Field quantities

Stress analysis:	Displacement field
Thermal Analysis:	Temperature or heat flux
Fluid Flow Analysis:	Velocity (potential) function

Mathematical explanation:

FE Method is a piecewise polynomial interpolation. It involves converting a governing set of complex partial differential equations into a much larger set of approximate simple algebraic equations.

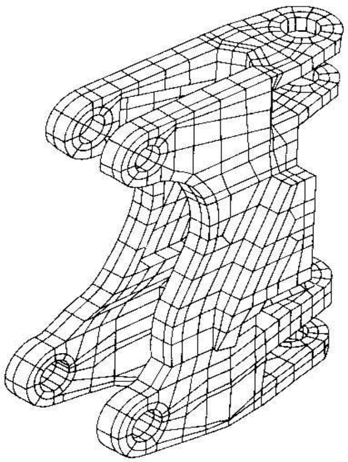


Figure 1–7 Three-dimensional solid element model of a swing casting for a backhoe frame

Within each element, a field quantity such as displacement is interpolated (as a polynomial) from values at the nodes. By connecting the elements together, the field quantity becomes interpolated over the entire structure (or continuum) in a piecewise fashion.

The "best" approximation for the field quantity is the one which minimizes the deviation from "exact" solution. This deviation is usually measured by potential energy functions.

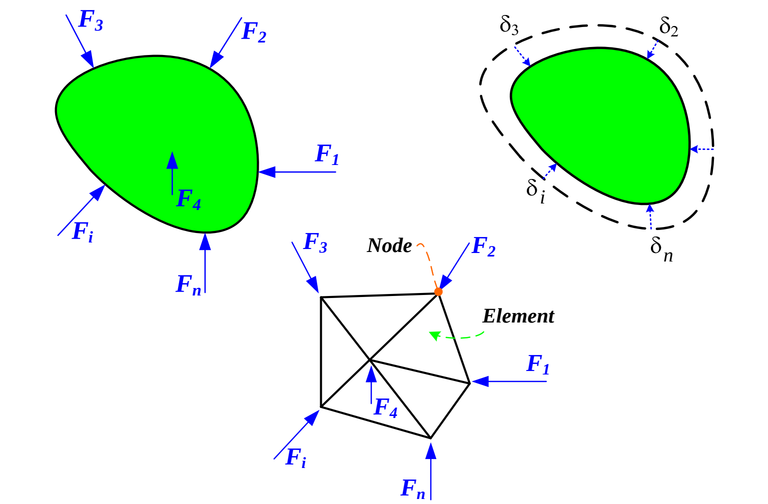
The displacement-based forms of these functions are called "strain-energy" functions.

Element Formulation Approach

Direct stiffness method
Potential Energy *
Weighted Residual (Galerkin)

The potential energy approach is equivalent to virtual work principle and can be stated as follows:

Assume a structure under application of forces is in a state of equilibrium. If the structure deforms due to a set of small "compatible" virtual displacements, the (virtual) work done by the external forces is equal to the (virtual) strain energy of the resulting internal stresses (or strains).



At the element level:

$\delta U_e = \delta W_e$

where

$\delta U_e \equiv$ Virtual strain energy

$\delta W_e \equiv$ Virtual work of external forces acting through virtual displacements

Finite Element solution involves selection of an "approximate" function to describe the displacement behavior within an element.

From displacement approximation, strain and stress are computed and the following expression is formed:

$$\int_{Vol} \{\sigma\}^T \{\varepsilon\} \, d\,Vol = \text{ Work done by all external forces}$$

where $\{\varepsilon\} = [\partial]\{u\}$ is the strain equation (per element)

$$[\partial] = \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix}$$

Combining the total strain energies and work of all elements, and equating work with energy, results in the following general set of simultaneous equations:

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

This equation can be solved to obtain the displacement values for all the points (nodes) which are vertices of the elements.

The obtained displacements can now be "back-substituted" into individual element's equilibrium equation to obtain element forces and, subsequently, stresses and strains.

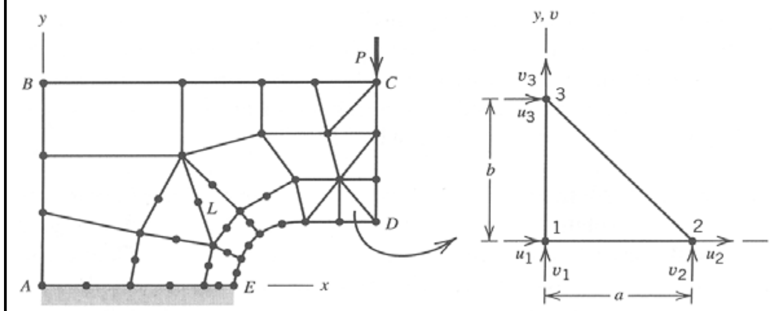
$[K]\{u\} = \{F\}$ → will be solved to obtain $\{u\}$ for each element. From displacement values, strain $\{\varepsilon\}$ within each element is computed. For example, in a one-dimensional (Rod) element:

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

Finally, stress is determined by:

$\{\sigma\} = [E]\ \{\varepsilon\}$ → determines the stress per element.

As an example, consider the general plane triangular element. The *x*-dir. and *y*-dir. displacement components of any given point inside the element are represented by u and v respectively (nodal displacements are indicated by corresponding subscripts).



Assuming a linear polynomial for approximation of displacement field:

$u = \beta_1 + \beta_2 x + \beta_3 y$

$v = \beta_4 + \beta_5 x + \beta_6 y$

Where β_i's are displacement field coefficients, also known as "*generalized coordinates*."

These coefficients are determined by substituting the coordinates (x and y) of the nodes and solving the resulting set of 6 simultaneous equations.

Boundary conditions:

@ $x = 0$ and $y = 0$ → $u = u_1$

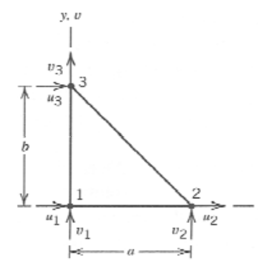
@ $x = 0$ and $y = 0$ → $v = v_1$

@ $x = a$ and $y = 0$ → $u = u_2$

@ $x = a$ and $y = 0$ → $v = v_2$

@ $x = 0$ and $y = b$ → $u = u_3$

@ $x = 0$ and $y = b$ → $v = v_3$



Substitute and solve to obtain:

$\beta_1 = u_1$ $\beta_2 = (u_2 - u_1) / a$ $\beta_3 = (u_3 - u_1) / b$

$\beta_4 = v_1$ $\beta_5 = (v_2 - v_1) / a$ $\beta_6 = (v_3 - v_1) / b$

Substituting for the coefficients in the polynomial equations:

$$u = (1 - \frac{x}{a} - \frac{y}{b})u_1 + \frac{x}{a}u_2 + \frac{y}{b}u_3 \qquad v = (1 - \frac{x}{a} - \frac{y}{b})v_1 + \frac{x}{a}v_2 + \frac{y}{b}v$$

Now we can obtain strains within the element by using the classical strain-displacement relations:

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad or \quad \varepsilon_x = \beta_3$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \quad or \quad \varepsilon_y = \beta_6$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad or \quad \gamma_{xy} = \beta_3 + \beta_5$$

We'll note that the strain terms do not depend on either x or y. Thus, the proposed linear displacement approximation results in a state of constant strain within the element!

Modeling the problem and checking the results

Modeling consists of simulation of a physical structure or a physical process by means of analytical or numerical constructs.

Therefore, it requires a good understanding of the problem that is to be modeled.

This understanding helps in determining the type of discretization suitable for a given problem, number of elements used in the model, approximation of physical boundary conditions, including loading and restraint, among others.

Discretization and other approximations

During FE study, we analyze a mathematical approximation of a physical problem. The approximation will invariably introduce *modeling error*.

As an example, the elementary beam theory ignores the transverse shear deformation. While this may be an excellent approximation for long slender beams, it is not valid for short non-slender ones.

If the beam to be modeled is deep, transverse shear deformation may become important and the element formulation should account for it. If the beam is very deep, then the elementary beam formulation may be altogether incorrect, and a two- or three-dimensional element formulation may be more appropriate.

User Responsibility

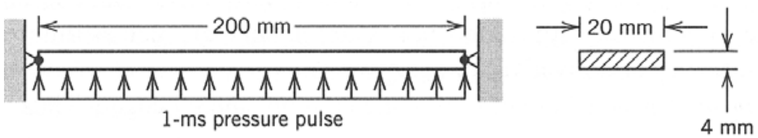
As the modern FE software has become easier to use, with colorful results contours, even inept users can produce some sort of an answer. However, results can be produced by both good and bad models.

Flawed FE models including poor mesh, inappropriate element types, incorrect loads, and improper support restraints may produce answers that appear reasonable upon casual inspection.

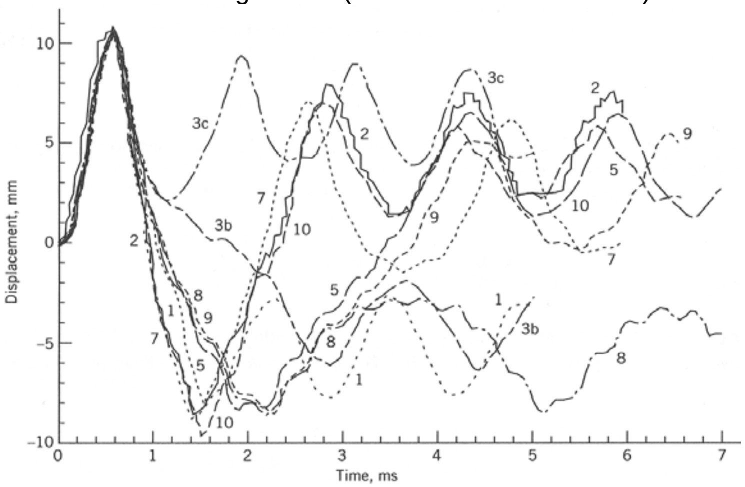
Understanding the physical nature of the problem as well as the behavior of the finite elements are important prerequisite to prepare the user well enough to undertake an FE-based analysis.

Example problem

Find the displacement response at the center of the beam



FE modeling results (based on different codes)



Basic building blocks in the Finite Element solution

1. **Discretization**
2. **Element formulation and interpolation**
3. **Assembly of elements**
4. **Constraint application**
5. **Global solution**
6. **Results extraction**