

Introduction to Finite Element Modeling

Engineering analysis of mechanical systems have been addressed by deriving differential equations relating the variables of through basic physical principles such as equilibrium, conservation of energy, conservation of mass, the laws of thermodynamics, Maxwell's equations and Newton's laws of motion. However, once formulated, solving the resulting mathematical models is often impossible, especially when the resulting models are non-linear partial differential equations. Only very simple problems of regular geometry such as a rectangular of a circle with the simplest boundary conditions were tractable.

The finite element method (FEM) is the dominant discretization technique in structural mechanics. The basic concept in the physical interpretation of the FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points.

The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements. The disconnection-assembly concept occurs naturally when examining many artificial and natural systems. For example, it is easy to visualize an engine, bridge, building, airplane, or skeleton as fabricated from simpler components. Unlike finite difference models, finite elements do not overlap in space.

Objectives of FEM in this Course

- Understand the fundamental ideas of the FEM
- Know the behavior and usage of each type of elements covered in this course
- Be able to prepare a suitable FE model for structural mechanical analysis problems
- Can interpret and evaluate the quality of the results (know the physics of the problems)
- Be aware of the limitations of the FEM (don't misuse the FEM - a numerical tool)

Finite Element Analysis

A typical finite element analysis on a software system requires the following information:

1. Nodal point spatial locations (geometry)
2. Elements connecting the nodal points
3. Mass properties
4. Boundary conditions or restraints
5. Loading or forcing function details
6. Analysis options

Because FEM is a discretization method, the number of degrees of freedom of a FEM model is necessarily finite. They are collected in a column vector called u . This vector is generally called the DOF vector or state vector. The term nodal displacement vector for u is reserved to mechanical applications.

FEM Solution Process

Procedures

1. Divide structure into pieces (elements with nodes) (discretization/meshing)
2. Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure (forming element matrices)
3. Solve the system of equations involving unknown quantities at the nodes (e.g., displacements)
4. Calculate desired quantities (e.g., strains and stresses) at selected elements

Basic Theory

The way finite element analysis obtains the temperatures, stresses, flows, or other desired unknown parameters in the finite element model are by minimizing an energy functional. An energy functional consists of all the energies associated with the particular finite element model. Based on the law of conservation of energy, the finite element energy functional must equal zero.

The finite element method obtains the correct solution for any finite element model by minimizing the energy functional. The minimum of the functional is found by setting the derivative of the functional with respect to the unknown grid point potential for zero.

Thus, the basic equation for finite element analysis is $\frac{\partial F}{\partial p} = 0$

where F is the energy functional and p is the unknown grid point potential (In mechanics, the potential is displacement.) to be calculated. This is based on the principle of virtual work, which states that if a particle is under equilibrium, under a set of a system of forces, then for any displacement, the virtual work is zero. Each finite element will have its own unique energy functional.

As an example, in stress analysis, the governing equations for a continuous rigid body can be obtained by minimizing the total potential energy of the system. The total

potential energy Π can be expressed as: $\Pi = \frac{1}{2} \int_{\Omega} \mathbf{s}^T \mathbf{e} dV - \int_{\Omega} d^T \mathbf{b} dV - \int_{\Gamma} d^T \mathbf{q} dS$ where \mathbf{s}

and \mathbf{e} are the vectors of the stress and strain components at any point, respectively, \mathbf{d} is the vector of displacement at any point, \mathbf{b} is the vector of body force components per unit volume, and \mathbf{q} is the vector of applied surface traction components at any surface point.

The volume and surface integrals are defined over the entire region of the structure Ω and that part of its boundary subject to load Γ . The first term on the right hand side of this

equation represents the internal strain energy and the second and third terms are, respectively, the potential energy contributions of the body force loads and distributed surface loads.

In the finite element displacement method, the displacement is assumed to have unknown values only at the nodal points, so that the variation within the element is described in terms of the nodal values by means of interpolation functions. Thus, within any one element, $\mathbf{d} = \mathbf{N} \mathbf{u}$ where \mathbf{N} is the matrix of interpolation functions termed shape functions and \mathbf{u} is the vector of *unknown nodal displacements*. (\mathbf{u} is equivalent to \mathbf{p} in the basic equation for finite element analysis.) The strains within the element can be expressed in terms of the element nodal displacements as $\mathbf{e} = \mathbf{B} \mathbf{u}$ where \mathbf{B} is the *strain displacement matrix*. Finally, the stresses may be related to the strains by use of an elasticity matrix (e.g., Young's modulus) as $\mathbf{s} = \mathbf{E} \mathbf{e}$.

The total potential energy of the discretized structure will be the sum of the energy contributions of each individual element. Thus, $\Pi = \sum_e \Pi_e$ where Π_e represents the total potential energy of an individual element.

$$\Pi_e = \frac{1}{2} \int_{\Omega_e} \mathbf{u}^T (\mathbf{B}^T \mathbf{E} \mathbf{B})^T \mathbf{u} dV - \int_{\Omega_e} \mathbf{u}^T \mathbf{N}^T \mathbf{p} dV - \int_{\Gamma} \mathbf{u}^T \mathbf{N}^T \mathbf{q} dS = 0.$$

Taking the derivative $\frac{\partial \Pi_e}{\partial \mathbf{u}} = \frac{1}{2} \int_{\Omega_e} (\mathbf{B}^T \mathbf{E} \mathbf{B})^T \mathbf{u} dV - \int_{\Omega_e} \mathbf{N}^T \mathbf{u} dV - \int_{\Gamma} \mathbf{N}^T \mathbf{q} dS = 0$ one gets the

element equilibrium equation $\mathbf{k} \mathbf{u} - \mathbf{f} = \mathbf{0}$ where $\mathbf{f} = \int_{\Omega_e} \mathbf{N}^T \mathbf{u} dV + \int_{\Gamma} \mathbf{N}^T \mathbf{q} dS$ and

$\mathbf{k} = \int_{\Omega_e} (\mathbf{B}^T \mathbf{E} \mathbf{B})^T \mathbf{u} dV$ and \mathbf{k} is known as the element stiffness matrix.

The physical significance of the vectors \mathbf{u} and \mathbf{f} varies according to the application being modeled.

Application Problem	State (DOF) vector \mathbf{d} represents	Forcing vector \mathbf{f} represents
Structures and solid mechanics	Displacement	Mechanical force
Heat conduction	Temperature	Heat flux
Acoustic fluid	Displacement potential	Particle velocity
Potential flows	Pressure	Particle velocity
General flows	Velocity	Fluxes
Electrostatics	Electric potential	Charge density
Magnetostatics	Magnetic potential	Magnetic intensity

Discretization

- Meshing

Coarse: Faster computation; not concerned about stress concentrations, singularities, or warping. Not near changes in geometry or displacement constraints or changes in material including thickness.

Fine: Best approximation but at the cost of the computation time.

Look for disproportionate stress level changes from node to node or plate to plate and large adjacent node displacement differences to determine if need to refine the mesh.

Nodes should be defined at locations where changes of geometry or loading occur. Changes in geometry relate to thickness, material and/or curvature.

A simple check, if you can, is to decrease the mesh size by 50%, re-run analysis, and compare the change of magnitude of stresses and strains. If there is no significant change, then ok.

In most companies, all of this knowledge of mesh size will be known and might be set a FEA control file.

- Degrees of Freedom

Constrain structure to prevent rigid body motion

Restrict motion in non-desireable directions

- Applied Forces

Static

Static distributed

Transient

Harmonic vibratory

- Element Types

Dictated by features of FEA software

1-D to 3-D

Pure stress, stress + bending, thermal, fluid

Triangular Element

model transitions between fine and coarse grids

Irregular structures

Warped surfaces

Quadrilateral element

Should lie on an exact plane; else moment on the membrane is produced.

3-D two-node truss

3 DOF, no bending loads

Stress constant over entire element

Typically used in two-force members ("RBAR")

3-D two-node beam

specify moments of inertia in both local X and local Y

6 DOF M_x M_y M_z

A typical beam

3-D 4-node plate

Modeling of structures where bending (out of plane) and/or membrane (in-plane) stress play equally important roles in the behavior of that particular structure

Each node has 6 DOF

Must specify plate thickness

2-D (3)4-node solid

"isoparametric four-node solid"

Common in 2-D stress problems and natural frequency analysis for solid structure.

"Thin" in that stress magnitude in third direction is considered constant over the element thickness.

2 translational DOF and no rotational DOF

2-D 2-node truss

No bending loads

uniaxial tension-compression

Straight bar with uniform properties from end to end

2 DOF, no rotation

2-D 2-node beam

3 DOF - translation x, y and rotation about z

Any cross section for which moment of inertia can be computed