Finite Element Analysis

Computer Aided Analysis & Design for Electrical Engineering

BE Electrical Semester-6

Gujarat Technological University

1. What is finite element method? Explain in brief.

- Finite Element Analysis is a numerical technique for finding approximate solutions of partial differential equations and integral equations.
- The finite element method is essentially based on subdivision the whole domain in a fixed number of sub-domains.
- This larger system problem is converted into smaller simpler parts called finite elements.
- The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem.
- FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis.
- It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm.
- FEA is a good choice for analyzing problems over complicated, when the domain changes, when the desired precision varies over the entire domain, or when the solution lacks smoothness.

2. Explain the advantages of finite element analysis of electrical machines.

- The finite element method (FEM) has proved to be particularly flexible, reliable and effective in the analysis and synthesis of power-frequency electromagnetic and electromechanical devices.

- Even in the hands of non-specialists, modern FEM packages are user friendly and allow for calculating the electromagnetic field distribution and integral parameters without detailed knowledge of applied mathematics.
- The FEM can analyze PM circuits of any shape and material.
- There is no need to calculate reluctances, leakage factors or the operating point on the recoil line.
- The PM demagnetization curve is input into the finite element program which can calculate the variation of the magnetic flux density throughout the PM system.
- An important advantage of finite element analysis over the analytical approach to PM motors is the inherent ability to calculate accurately armature reaction effects, inductances and the electromagnetic torque variation with rotor position.
- The verification of analytical design can be carried out with help of finite element analysis and improvement in the design can be obtained.

3. Explain the basic requirements of mesh generation in finite element analysis.

- The accuracy of the finite element solution is dependent on the mesh topology. The mesh is thus an important part of any finite element model and attention should be placed on creating it.
- Essentially, there are two types of mesh generators.
- The first being an **analytical mesh generator** that defines the problem geometry using large global elements. These global elements are subsequently refined according to the user, usually automatically.

- The other type of mesh generator is a **synthetic generator** where the user designs a mesh region at a node-by-node level and the model is the union of a number of different mesh regions.
- Modern finite element packages can generate a mesh automatically from the geometric outline of the problem drawn in a CAD type package.
- These mesh generators usually construct the mesh using a Delaunay triangulation method. Automating the mesh generation drastically reduces the manpower costs.
- Fully automated mesh generation can only be achieved if the errors which arise from the mesh discretization are taken into account.
- This is called self-adaptive meshing, and relies on an accurate and reliable method of estimating the discretization errors in the mesh.
- Modern finite element packages that use self-adaptive meshing normally calculate the discretization error estimates from a finite element solution.
- These packages usually create a crude mesh which is solved. Error estimates are made from this solution and the mesh is refined in the approximate places.
- This process is repeated until the required level of accuracy is obtained for the model.
- The error estimates used in the program depend generally on the application, but general finite element programs usually calculate their error estimates from the change in flux density across element edges.

4. Explain boundary conditions applied in electromagnetic field problems.

- Natural boundaries very rarely exist in electromagnetic field problems. In most types of applications, the electromagnetic field is an infinitely extending space.
- In these applications boundaries are used to simplify the finite element model and approximate the magnetic vector potential at node points. Boundaries due to symmetry greatly reduce the size of the finite element model.
- Boundary conditions can be classified into three types
    - **Dirichlet boundary conditions**

      Dirichlet boundary conditions require the magnetic vector potential, at a particular point, to take on a prescribed value,

      $$\vec{A} = m$$

      where m is the specified value.

      Dirichlet boundaries force the flux lines to be parallel to the boundary's edge.

      The high relative permeability of the ferromagnetic yoke would ensure that most of the flux remains inside the yoke and in most motor designs this boundary condition is a reasonable simplification to make.
    - **Neumann boundary conditions**

      Neumann boundary conditions require the normal derivative of the magnetic vector potential be zero

      $$\frac{\partial A}{\partial n} = 0$$

This type of boundary is not satisfied exactly by the finite element solution but only the mean value of the boundary. Neumann boundaries are natural boundaries in the FEM since they do not have to be specified explicitly. Flux lines cross a Neumann boundary orthogonally. Neumann boundaries are used mainly in symmetry problems where the flux is orthogonal to a plane. This only occurs in no-load operation in most electrical motors.

- **Interconnection boundary conditions**

  Interconnection boundary conditions set the constraint between two nodes. This could be between two geometrically adjacent nodes or between two nodes at a particular interval apart. This type of boundary has only one of the two potentials independently specified and is still exactly satisfied in the finite element solution.

  The relationship between the two nodes is

$$\overrightarrow{A_m} = a\overrightarrow{A_n} + b$$

  where a and b are factors which link the two nodes.

5. Explain in brief the procedure for finite element analysis.

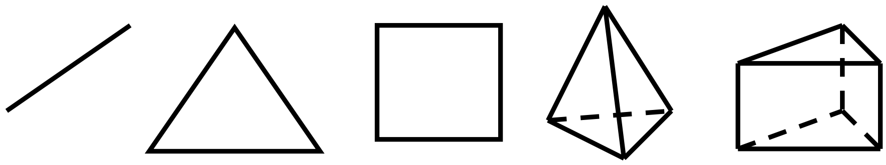
Finite Element Analysis is organized as

- **Partition of the domain**

  Dividing the domain of the problem into a collection of subdomains, with each subdomain represented by a set of element equations to the original problem.

  - Accurate representation of complex geometry

- Inclusion of dissimilar material properties
- Easy representation of the total solution
- Capture of local effects



- **Choice of interpolating function**

  The element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are often partial differential equations (PDE). In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE.

- **Formulation of the system**

  systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

- **Solution of the problem**

  From the defined interpolating function and formulation of the system the analysis is started. According to interpolating function and system the equations are solved. The result of the equations is observed and compared with the desired result.

6. Explain the steps involved in finite element software packages.

The following are the components of FEA packages:

- **Pre-processor**

  Pre-processor is a module where the finite element model is created by the user. This module allows new models to be created and old models to be altered. The different parts of any finite element model are

  - **Drawing**

    Drawing the geometric outline of the model using graphical drawing tools much like any CAD package including mirroring and copying features.

  - **Material**

    The different regions of the geometry model are assigned magnetic material properties. The materials can have linear or nonlinear magnetic characteristics. Each material can be defined a particular conductivity. For PM materials the coercivity of the material is also defined.

  - **Electric circuits**

    Regions which contain coils are linked to current or voltage sources. The user specifies the number of turns per coil and the current magnitude.

  - **Constrains**

    The edges of the model usually need constraints and this is done by defining constraints graphically. Periodic constraints are easily defined, by the user, using graphical mouse clicks.

Once all the components of the model are described the finite element model can be solved.

- **Solver**

  Solver module solves numerically the field equations.

  - **Type of solution**

    The pre-processor module sets which solver (electrostatic, magnetostatic, eddy-current) should be used.

  - **Meshing**

    Mostly, adaptive meshing is implemented to ensure efficient mesh discretization. The solver starts by creating a coarse finite element mesh and solving it.

  - **Error estimation**

    An error estimate is produced from this solution, and the mesh is refined and solved again. This is repeated until the mesh is refined sufficiently to produce an accurate result.

- **Post-processor**

  Post-processor is an interactive module that displays field quantities such as magnetic vector potential, flux density, field intensity and permeability. It also gives the user access to a vast amount of information regarding the finite element solution such as energy, force, torque and inductance, which are all built into the module. The results are shown in the form of **Graphical plots** and **Numerical results.**