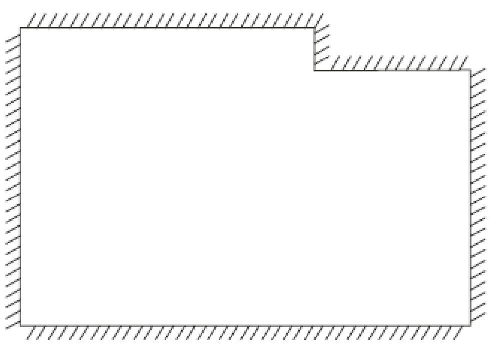
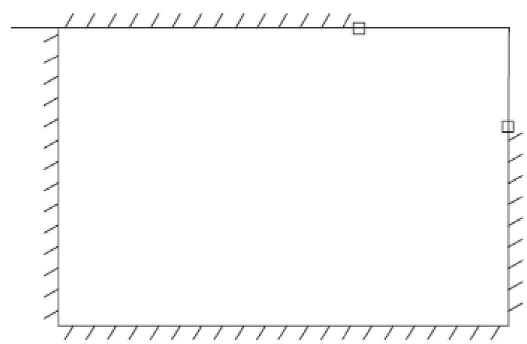
Finite Element Method Vs Classical Methiods

1. In classical methods exact equations are formed and exact solutions are obtained where as in finite element analysis exact equations are formed but approximate solutions are obtained.
2. Solutions have been obtained for few standard cases by classical methods, where as solutions can be obtained for all problems by finite element analysis.
3. Whenever the following complexities are faced, classical method makes the drastic assumptions' and looks for the solutions:
   (a) Shape
   (b) Boundary conditions
   (c) Loading

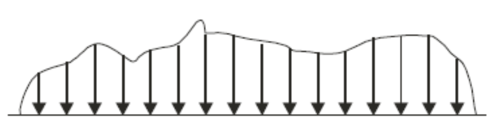
Figure below shows such cases in the analysis of slabs (plates).



(a) Irregular shaper



(b) Irregular boundary condition



(c) Irregular loading

To get the solution in the above cases, rectangular shapes, same boundary condition along a side and regular equivalent loads are to be assumed. In FEM no such assumptions are made. The problem is treated as it is.

4. When material property is not isotropic, solutions for the problems become very difficult in classical method. Only few simple cases have been tried successfully by

researchers. FEM can handle structures with anisotropic properties also without any difficulty.

5. If structure consists of more than one material, it is difficult to use classical method, but finite element can be used without any difficulty.
6. Problems with material and geometric non-linearities can not be handled by classical methods. There is no difficulty in FEM.

Hence FEM is superior to the classical methods only for the problems involving a number of complexities which cannot be handled by classical methods without making drastic assumptions. For all regular problems, the solutions by classical methods are the best solutions. Infact, to check the validity of the FEM programs developed, the FEM solutions are compared with the solutions by classical methods for standard problems.

NEED FOR STUDYDING FINITE ELEMENT METHOD

Now, a number of users' friendly packages are available in the market. Hence one may ask the question ***'What is the need to study FEA?'***

The above argument is not sound. The finite element knowledge makes a good engineer better while just user without the knowledge of FEA may produce more dangerous results. To use the FEA packages properly, the user must know the following points clearly:

1. Which elements are to be used for solving the problem in hand?
2. How to discritise to get good results.
3. How to introduce boundary conditions properly.
4. How the element properties are developed and what are their limitations.
5. How the displays are developed in pre and post processor to understand their limitations.
6. To understand the difficulties involved in the development of FEA programs and hence the need for checking the commercially available packages with the results of standard cases.

Unless user has the background of FEA, he may produce worst results and may go with overconfidence. Hence it is necessary that the users of FEA package should have sound knowledge of FEA.

Warnings by Commercial FEM Software Packages

When hand calculations are made, the designer always gets the feel of the structure and gets rough idea about the expected results. This aspect cannot be ignored by any designer, whatever is the reliability of the program.

User must remember that structural behavior is not dictated by the computer programs. Hence the designer should develop feel of the structure and make use of the programs to get numerical results which are close to structural behavior.

GENERAL DESCRIPTION OF FINITE ELEMENT METHOD

In engineering problems there are some basic unknowns. If they are found, the behavior of the entire structure can be predicted. The **basic unknowns** or the **Field variables** which are encountered in the engineering problems are:

- **Displacements in solid mechanics**,
- **Velocities in fluid mechanics**,
- **Electric and magnetic potentials in electrical engineering** and
- **Temperatures in heat flow problems.**

In a continuum, these unknowns are infinite. The finite element procedure reduces such unknowns to a finite number by dividing the solution region into small parts called **elements** and by expressing the unknown field variables in terms of assumed **approximating functions (Interpolating functions/Shape functions)** within each element. The approximating functions are defined in terms of field variables of specified points called **nodes** or **nodal points**. Thus in the finite element analysis the unknowns are the field variables of the nodal points. Once these are found the field variables at any point can be found by using interpolation functions.

GENERAL STEPS OF THE FINITE ELEMENT METHOD

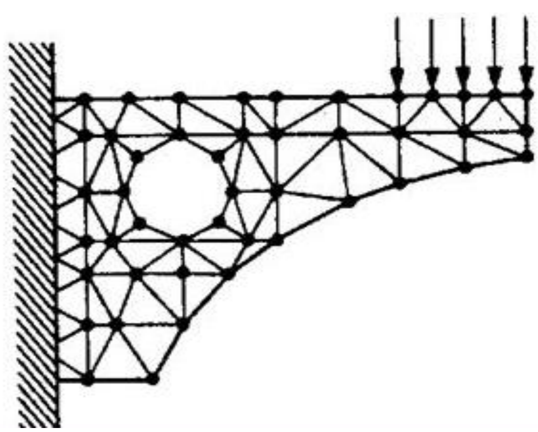
Step 0: Obtain a basic understanding of the problem you are attempting to solve.

- Are any classical solutions (closed-form) available?
- Is there any experimental solutions?
- Which modes of deformation do you expect to significantly contribute to the structure's behavior?

Step 1: Discretize and Select the Element Types

Step 1 involves dividing the body into an equivalent system of finite elements with associated nodes and choosing the most appropriate element type to model most closely the actual physical behavior.

- The total number of elements used and their variation in size and type within a given body are primarily matters of engineering judgment.
- The elements must be made **small enough to give usable results and yet large enough to reduce computational effort.**
- **Small elements** (and possibly higher order elements) are generally desirable where the results are changing rapidly, such as where changes in geometry occur; **large elements** can be used where results are relatively constant.
- **The choice of elements** used in a finite element analysis depends on the physical makeup of the body under actual loading conditions and on how close to the actual behavior the analyst wants the results to be. Judgment concerning the appropriateness of one-, two-, or three-dimensional idealizations is necessary. Moreover, the choice of the most appropriate element for a particular problem is one of the major tasks that must be carried out by the designer/analyst.



- Elements that are commonly employed in practice are:
- The **primary line elements consist of bar** (or truss) and **beam elements.** They have a cross-sectional area but are usually represented by line segments. In general, the cross-sectional area within the element can vary. These elements are often used to model trusses and frame structures. The simplest line element (called a linear element) has two nodes, one at each end, although higher-order elements having three nodes or more (called quadratic, cubic, etc. elements) also exist.

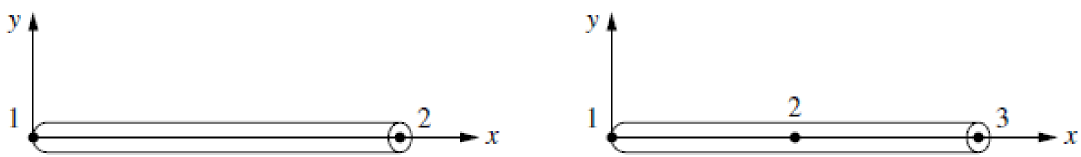


Figure: Simple two-noded line element (typically used to represent a bar or beam element) and the higher-order line element

- The **basic two-dimensional (or plane) elements** are loaded by forces in their own plane (plane stress or plane strain conditions). They are triangular or quadrilateral elements. The simplest two-dimensional elements have corner nodes only (linear elements) with straight sides or boundaries although there are also higher-order elements, typically with midside nodes (called quadratic elements).

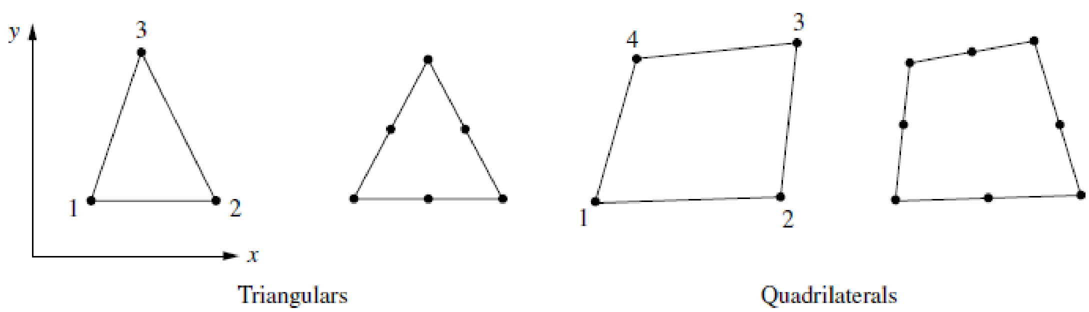


Figure: Simple two-dimensional elements with corner nodes (typically used to represent plane stress/strain) and higher-order two-dimensional elements with intermediate nodes along the sides.

- The **most common three-dimensional elements** are **tetrahedral and hexahedral** (or brick) elements; they are used when it becomes necessary to perform a three-dimensional stress analysis. The basic three-dimensional elements have corner nodes only and straight sides, whereas higher-order elements with midedge nodes (and possible midface nodes) have curved surfaces for their sides.

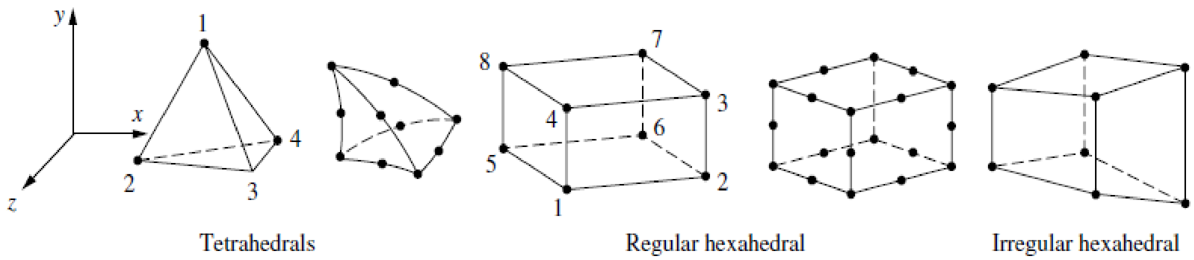


Figure: Simple three-dimensional elements (typically used to represent three-dimensional stress state) and higher-order three-dimensional elements with intermediate nodes along edges.

- The **axisymmetric element** is developed by rotating a triangle or quadrilateral about a fixed axis located in the plane of the element through 360. This element can be used when the geometry and loading of the problem are axisymmetric.

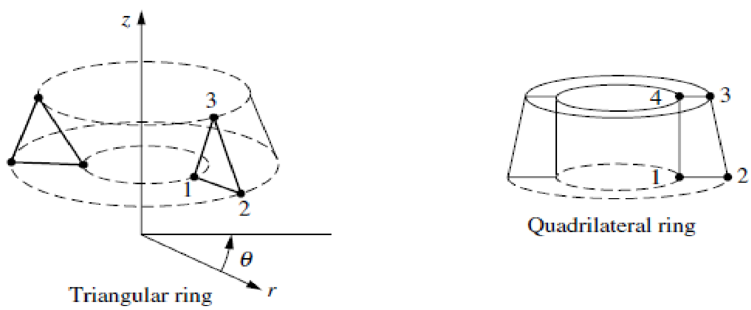


Figure: Simple axisymmetric triangular and quadrilateral elements used for axisymmetric problems.

Step 2 - Select a Displacement Function

Step 2 involves choosing a **displacement function** within each element. The function is defined within the element using the nodal values of the element. **Linear, quadratic, and cubic polynomials** are frequently used functions because they are simple to work with in finite element formulation. The functions are expressed in terms of the nodal unknowns (in the two-dimensional problem, in terms of an x and a y component). Hence, the finite element method is one in which a continuous quantity, such as the displacement throughout the body, is approximated by a discrete model composed of a set of **piecewise-continuous functions** defined within each finite domain or finite element.

Step 3 - Define the Strain/Displacement and Stress/Strain Relationships

Strain/displacement and stress-strain relationships are necessary for deriving the equations for each finite element. For one-dimensional small strain deformation, say, in the x direction, we have strain ε_x, related to displacement u (kinematics relationship) by:

$$\varepsilon_x = \frac{du}{dx}$$

In addition, the stresses must be related to the strains through the stress strain law (generally called the constitutive law). The ability to define the material behavior accurately is most important in obtaining acceptable results. The simplest of stress-strain laws, Hooke's law, often used in stress analysis (constitutive relationship), is given by:

$$\sigma_x = E\varepsilon_x$$

Step 4 Derive the Element Stiffness Matrix and Equations

There are three standard methods for deriving the element stiffness matrix:

a) **Direct (Equilibrium) Methods**:

Based on physical reasoning and limited to simple element types and simple **kinematics** and **constitutive relationships**. According to this method, the stiffness matrix and element equations relating nodal forces to nodal displacements are obtained using force **equilibrium conditions for a basic element**, **along with force deformation relationships**. This method is most easily adaptable to line or one dimensional elements (spring, bar, and beam elements).

b) **Work or Energy (Variation Methods)**

It uses the fact that a deformable body under loading and restraint conditions will assume a configuration that minimizes the potential energy of the total system.

To develop the stiffness matrix and equations for two- and three-dimensional elements, it is much easier to apply a work or energy method. The **principle of virtual work** (using virtual displacements), the **principle of minimum potential energy**, and **Castigliano's theorem** are methods frequently used for the purpose of derivation of element equations.

c) **Weighted Residual Methods** – Stiffness relationship results from a mathematical approach to obtain approximate solutions to the governing PDE's. Approximate solutions of differential equations satisfy only part of the conditions of the problem: the differential equation may be satisfied only at a few positions, rather than at each point. The weighted residual methods allow the finite element method to be applied directly to any differential equation.

Using any of the methods just outlined will produce the equations to describe the behavior of an element. These equations are written conveniently in matrix form as:

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ \vdots \\ f_n \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & \dots & k_{1n} \\ k_{21} & k_{22} & k_{23} & \dots & k_{2n} \\ k_{31} & k_{32} & k_{33} & \dots & k_{3n} \\ \vdots & & & & \vdots \\ k_{n1} & & & \dots & k_{nn} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ \vdots \\ d_n \end{Bmatrix}$$

Or in compact matrix form as

$$\{f\} = [k]\{d\}$$

Where $\{f\}$ is the vector of element nodal forces, $[k]$ is the element stiffness matrix (normally square and symmetric), and $\{d\}$ is the vector of unknown element nodal degrees of freedom or generalized displacements, n. Here generalized displacements may include such quantities as actual displacements, slopes, or even curvatures.

Step 5 Assemble the Element Equations to Obtain the Global or Total Equations and Introduce Boundary Conditions

In this step the individual element nodal equilibrium equations generated in step 4 are assembled into the global nodal equilibrium equations. Another more direct method of superposition (called the **Direct Stiffness Method**), whose basis is nodal force equilibrium, can be used to obtain the global equations for the whole structure. Implicit in the direct stiffness method is the concept of continuity, or compatibility, which requires that the structure remain together and that no tears occur anywhere within the structure. The final assembled or global equation written in matrix form is

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

Where $\{F\}$ is the vector of global nodal forces, $\{K\}$ is the structure global or total stiffness matrix, (for most problems, the global stiffness matrix is square and symmetric) and $\{d\}$ is now the vector of known and unknown structure nodal degrees of freedom or generalized displacements. It can be shown that at this stage, the global stiffness matrix $\{K\}$ is a singular matrix because its determinant is equal to zero. To remove this singularity problem, we must invoke certain boundary conditions (or constraints or supports) so that the structure remains in place instead of moving as a rigid body.

<u>Step 6 - Solve for the Unknown Degrees of Freedom (or Generalized Displacements)</u>

Once the element equations are assembled and modified to account for the boundary conditions, a set of simultaneous algebraic equations that can be written in expanded matrix form as:

$$\begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} \\ K_{21} & K_{22} & \dots & K_{2n} \\ \vdots & & & \vdots \\ K_{n1} & K_{n2} & \dots & K_{nn} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ \vdots \\ d_n \end{Bmatrix}$$

Where n is the structure total number of unknown nodal degrees of freedom. These equations can be solved for the d's by using an elimination method (such as Gauss's method) or an iterative method (such as Gauss Seidel's method).

<u>Step 7 Solve for the Element Strains and Stresses</u>

For the structural stress-analysis problem, important secondary quantities of strain and stress (or moment and shear force) can be obtained because they can be directly expressed in terms of the displacements determined in step 6.

Step 8 Interpret the Results

The final goal is to interpret and analyze the results for use in the design/analysis process. Determination of locations in the structure where large deformations and large stresses occur is generally important in making design/analysis decisions. Postprocessor computer programs help the user to interpret the results by displaying them in graphical form.