

# An Overview of the Finite Element Analysis

---

---

## 1.1 Introduction

Finite element analysis (FEA) involves solution of engineering problems using computers. Engineering structures that have complex geometry and loads, are either very difficult to analyze or have no theoretical solution. However, in FEA, a structure of this type can be easily analyzed. Commercial FEA programs, written so that a user can solve a complex engineering problems without knowing the governing equations or the mathematics; the user is required only to know the geometry of the structure and its boundary conditions. FEA software provides a complete solution including deflections, stresses, reactions, etc.

In order to become a skillful FEA user, a thorough understanding of techniques for modeling a structure, the boundary conditions and, the limitations of the procedure, are very crucial. Engineering structures, e.g., bridge, aircraft wing, high-rise buildings, etc., are examples of complex structures that are extremely difficult to analyze by classical theory. But FEA technique facilitates an easier and a more accurate analysis. In this technique the structure is divided into very small but finite size elements (hence the name finite element analysis). Individual behavior of these elements is known and, based on this knowledge; behavior of the entire structure is determined.

FEA solution of engineering problems, such as finding deflections and stresses in a structure, requires three steps:

1. Pre-process or modeling the structure
2. Analysis
3. Post processing

A brief description of each of these steps follows.

### **Step1: Pre-process or modeling the structure**

Using a CAD program that either comes with the FEA software or provided by another software vendor, the structure is modeled. The final FEA model consists of several elements that collectively represent the entire structure. The elements not only represent segments of the structure, they also simulate it's mechanical behavior and properties.

Regions where geometry is complex (curves, notches, holes, etc.) require increased number of elements to accurately represent the shape; where as, the regions with simple geometry can be represented by coarser mesh (or fewer elements). The selection of proper elements requires prior experience with FEA, knowledge of structure's behavior, available elements in the software and their characteristics, etc. The elements are joined at the nodes, or common points.

In the pre-processor phase, along with the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called “mesh”.

### Step 2: Analysis

In this step, the geometry, constraints, mechanical properties and loads are applied to generate matrix equations for each element, which are then assembled to generate a global matrix equation of the structure. The form of the individual equations, as well as the structural equation is always,

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

Where

$$\begin{aligned}\{F\} &= \text{External force matrix.} \\ [K] &= \text{Global stiffness matrix} \\ \{u\} &= \text{Displacement matrix}\end{aligned}$$

The equation is then solved for deflections. Using the deflection values, strain, stress, and reactions are calculated. All the results are stored and can be used to create graphic plots and charts in the post analysis.

### Step 3: Post processing

This is the last step in a finite element analysis. Results obtained in step 2 are usually in the form of raw data and difficult to interpret. In post analysis, a CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation, etc. A graphical representation of the results is very useful in understanding behavior of the structure

## 1.2 History of FEA

Engineering applications of finite element analysis is approximately 40 years old. Evolution of FEA is tied with the development in computer technology. With the enhancement in computer speed and storage capacity, FEA has become a very valuable engineering tool. NASA is credited for developing comprehensive FEA software in 1960's, known as NASTRAN. Rights of the software were purchased by McNeal Schwendler Corporation, who refined it and commercially marketed it under the name, MSC-NASTRAN. The first college course in FEA was offered in 1970. In the

early 1970's, application of FEA was limited to large corporations, who can afford expensive mainframe computers. However, in 1980's, with the introduction of desktop computers, application of FEA became popular and indispensable engineering tool. In late 80's, almost all the major FEA vendors introduced their software that can run on a PC.

In the past ten years, there were several significant development in FEA, including:

- Introduction of P- elements.
- Integration of sensitivity analysis and optimization capabilities.
- Availability of faster and cheaper desktop computers to run FEA software that previously required mainframe computers.
- Development of powerful CAD programs for modeling complex structures.
- Making software user-friendly.

### 1.3 How FEA works – Within software

The following steps can summarize FEA procedure that works inside software:

- Using the user's input, the given structure is graphically divided into small elements (sections or regions) so that each and every element's mechanical behavior can be defined by a set of differential equations.
- The differential equations are converted into algebraic equation, and then into matrix equations, suitable for a computer-aided solution.
- The element equations are combined and a global structural equation is obtained.
- Appropriate load and boundary conditions, supplied by the user, are incorporated in to the structural matrix.
- The structural matrix is solved and deflections of all the nodes are calculated.
- A node can be shared by several elements and the deflection at the shared node represents deflection of the sharing elements at the location of the node.
- Deflection at any other point in the element is calculated by interpolation of all the node points in the element.
- An element can have a linear or higher order interpolation function.
- The individual element matrix equations are assembled into a combined structure equation of the form  $\{F\}=[k]\{u\}$ .

As defined earlier,

$\{F\}$  = Column matrix of the externally applied loads.

$[k]$  = Stiffness matrix of the structure, which is always a symmetric matrix. This matrix is analogues to an equivalent spring constant of several connected springs.

$\{u\}$  = Column matrix representing the deflection of all the node points, that results when the load  $\{F\}$  is applied.

## 1.4 How FEA works – User’s interaction

The above described software procedure is mostly transparent to the user. A user has the following interaction with the software, through user’s computer.

- Create the geometry, representing the structure: A CAD modeling software is used to create the structure’s geometry.
- Provide the material properties, loads, constraints, etc.
- Analyze the result data.

## 1.5 Convergence – Assuring Optimum Mesh Size

How do we determine the exact number of elements for a structure and make sure that the FEA mesh is optimum? There is no exact answer to this question; however, if we keep refining a mesh until the variation in the result is less than a specified value, we will reach the desirable mesh density. Convergence refers to this process, where we optimize the mesh to arrive at the desired results. In general, there are three types of convergences:

1. **Von-Mises Stress (VMS) convergence:** Mesh is refined until the percentage variation in VMS is less than 1, 5, 10 or any given value selected by the user. VMS convergence should be avoided if there are stress concentration points, convergence will be difficult to achieve.
2. **Strain Energy Convergence:** Mesh is refined until the percentage variation in the average strain of elements is less than a chosen value. Strain convergence is a better criterion for optimizing an FEA mesh. Stress concentrations points do not significantly influence the average strain energy of elements and variation in strain energy is influenced by mesh size or polynomial order of the elements only.
3. **Deflection Convergence:** It is similar to the above convergences, except, node deflection values are used for the convergence criterion.

## 1.6 H- versus P- elements

In FEA, there are two types of elements:

1. H-elements and,
2. P-elements

H-element is the original and “classic” element. The name is derived from the field of numerical analysis, where the letter ‘h’ is used for the step size, to achieve convergence in the analysis. The h-element is always of low order, usually, linear or quadratic. When a finite element mesh is refined to achieve convergence, the procedure is called h-convergence. For h-elements, convergence is accomplished at the expense of excessively

large number of elements. The high stress concentration regions require a very fine mesh, thereby increasing the number of elements. Finite elements used by commercial programs in the 1970s and 80s, were all h-elements. However, with improvement in computer power and efficiency, a much more useful, p-elements were developed.

P-elements are relatively new, developed in late 1980s and offer not only the traditional static analysis, they provides option of optimizing a structure. P-elements can have edge-polynomial as high as 9<sup>th</sup> order, unlike the low order polynomials of h-elements. The high polynomial edge order of p-elements makes it possible to model a curved edge of a structure with accuracy. Therefore, fewer elements can be used to achieve convergence. In FEA, the number of elements in the mesh usually remains fixed; convergence is achieved by increasing the polynomial order of the p-elements, rather than refinement of the mesh. For optimization, as the dimensions of the structure being analyzed are changed, the number of elements remains constant. Only, the polynomial order of the elements is changed as needed.

## 1.6 Bottom-up and Top-down approach

When modeling a structure (creating an FEA model), bottom – up approach refers to creation of model by defining the geometry of the structure with nodes and elements. These nodes and elements represent the physical structure. When an FEA model is created by this procedure, it is known as a bottom-up approach. This is the original procedure for creating FEA mesh, and requires a substantial investment in time and skill. When this method is employed, most of analyst’s time is devoted to creation of the mesh, and only a fraction of time is spent for analysis and results interpretation.

In FEA, a top-down procedure refers to creation of FEA mesh by first building a solid model, using a 3-D CAD program, and then dividing the model into nodes and elements. Thus, the top-down method requires building of a geometric model of the structure and then using it to create an FEA mesh. The advantages of the top-down approach are obvious; we don’t have to define the geometry of individual elements in the structure, which can be very time consuming. Obviously, a 3-D model requires high-end computer hardware, along with familiarity with the modeling software.

## 1.7 Discretization or Division of a structure into small elements

In FEA, an engineering structure is divided into small elements. These elements coincide with the geometry of the structure and represent the geometry and the mechanical properties in the regions.

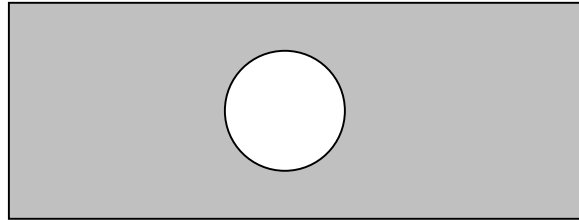
Selection of elements to represent the structure is a matter of engineering judgement and prior experience with FEA procedure. A sound advice for beginners is: keep the elements size small enough to yield good results and yet large enough to reduce computational time. Smaller elements are desirable where the results are changing rapidly (change in

geometry, sharp corners, etc.). Large elements can be used where the results (deflection or stresses) are relatively constant.

In FEA, discretization of a structural model is another name for mesh generation. Most of the commercial FEA programs have the capability of automatically generating FEA mesh. User has to provide the element type, mechanical properties, constraints and loads.

## 1.8 Element types

Let us assume that we wish to find stress concentration in a steel plate with holes. For the FEA analysis of this plate, we would need elements that have shapes of triangular plates, quadrilateral plates, and plates with curved edge. Then these elements can replace and represent each and every part of the plate, including the circular edges near the hole.



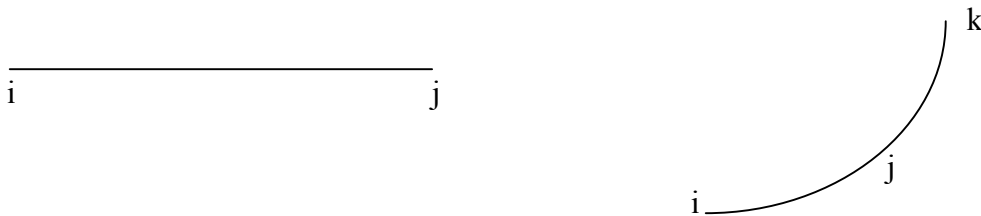
*Plate with a hole*

Thus, we need elements that have geometric shape similar to the real structure or region of the structure that is being modeled. One geometric shape cannot represent all possible engineering structural shapes. Therefore, we need elements that look like a plate, beam, cylinder, sphere, etc. However, in FEA, almost all structures can be approximated by the following basic elements:

1. **Line elements:** Element consisting of two nodes.

Example: Truss and beam elements.

In computers, a line, connecting two nodes at its ends as shown, represents a line element. The cross-sectional area is assumed constant throughout the element.

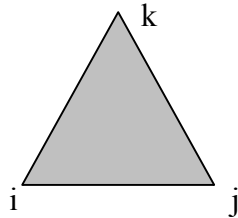


The element can have more than two nodes, and can be a curved rather than a straight line.

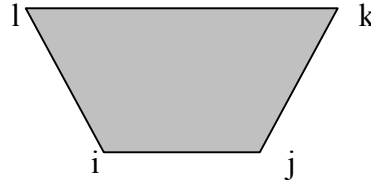
2. **2-D solid elements:** Elements that have geometry similar to a flat plate.

Example: Plane stress, plain strain, plates, shells, and axisymmetric elements.

2-D solid elements are plane elements, with constant thickness, and have either a triangular or quadrilateral shape, with 3 nodes or 4 nodes as shown.



*2-D Solid: Triangular*



*2-D Solid: Quadrilateral*

For higher order 2-D elements, the number of nodes can vary. For example, the element edges can be quadratic with 3 nodes on each edge. However, in most FEA analysis, only the straightedge elements are used.

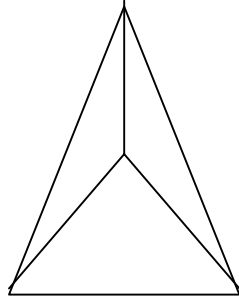
Loads on 2-D solid elements can be applied only in its plane, and deflections also occur only in the plane of the elements.

Axisymmetric element is a special case of 2-D plane stress element. We will discuss this element in detail later on.

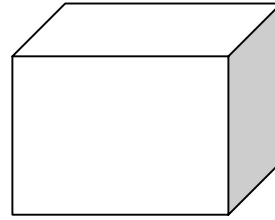
3. **3-D solid elements:** Element that have a 3-D geometry.

Example: Tetrahedron and hexahedron elements.

The basic 3-D solid elements have either a tetrahedral (4 faces) or hexahedral (6 faces) shape, as shown.



*Tetrahedral - 4-nodes*



*Hexahedral - 8-nodes*

The basic elements have corner nodes and straight edges, but the number of nodes and edge geometry can vary.

**NOTES**

- 1 For an accurate analysis in FEA, selection of the proper elements is very important. The selected elements must represent the engineering structure as close to the original structure as possible.
- 2 In addition to these basic elements, there are some special application elements, e.g., mass element and contact element. Almost all other special purpose elements can be derived from the three basic groups of the elements described above.