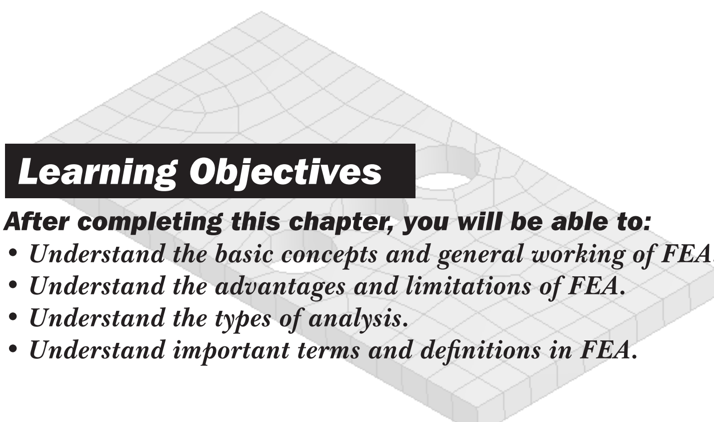
Chapter 1

Introduction to FEA



Learning Objectives

After completing this chapter, you will be able to:

- *Understand the basic concepts and general working of FEA.*
- *Understand the advantages and limitations of FEA.*
- *Understand the types of analysis.*
- *Understand important terms and definitions in FEA.*

INTRODUCTION TO FEA

The finite element analysis (FEA) is a computing technique that is used to obtain approximate solutions to boundary value problems. It uses a numerical method called finite element method (FEM). FEA involves the computer model of a design that is loaded and analyzed for specific results, such as stress, deformation, deflection, natural frequencies, mode shapes, temperature distributions, and so on.

The concept of FEA can be explained through a basic example involving measurement of the perimeter of a circle. To measure the perimeter of a circle without using the conventional formula, divide the circle into equal segments, as shown in Figure 1-1. Next, join the start point and the endpoint of each of these segments by a straight line. Now, you can measure the length of straight line very easily, and thus, the perimeter of the circle by adding the length of these straight lines.

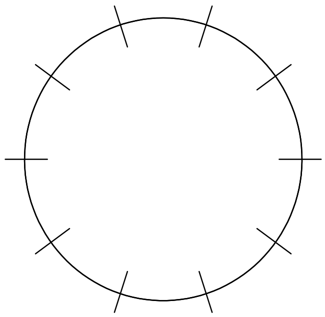


Figure 1-1 *The circle divided into small equal segments*

If you divide the circle into four segments only, you will not get accurate results. For accuracy, divide the circle into more number of segments. However, with more segments, the time required for getting the accurate result will also increase. The same concept can be applied to FEA also, and therefore, there is always a compromise between accuracy and speed while using this method. This compromise between accuracy and speed makes it an approximate method.

The FEA was first developed to be used in the aerospace and nuclear industries, where the safety of structures is critical. Today, even the simplest of products rely on FEA for design evaluation.

The FEA simulates the loading conditions of a design and determines the design response in those conditions. It can be used in new product design as well as in existing product refinement. A model is divided into a finite number of regions/divisions called elements. These elements can be of predefined shapes, such as triangular, quadrilateral, hexahedron, tetrahedron, and so on. The predefined shape of an element helps define the equations that describe how the element will respond to certain loads. The sum of the responses of all elements in a model gives the total response of the complete model.

General Working of FEA

A better knowledge of FEA helps in building more accurate models. Also, it helps in understanding the backend working of ANSYS. Here, a simple model is discussed to give you a brief overview of the working of FEA.

Figure 1-2 shows a spring assembly that represents a simple two-spring element model. In this model, two springs are connected in series and one of the springs is fixed at the left most endpoint, refer to Figure 1-2. In this figure, the stiffness of the springs has been represented by the spring constants K_1 and K_2. The movement of endpoints of each spring is restricted to the X direction only. The change in position from the undeformed state of each endpoint can be defined by the variables X_1 and X_2. The two forces acting on the end points of the springs are represented by F_1 and F_2.

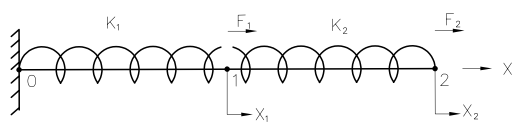


Figure 1-2 *Representation of a two-spring assembly*

To develop a model that can predict the state of this spring assembly, you can use the linear spring equation given below:

$$F = KX$$

where,
F = force applied,
X = displacement, and
K = spring constant

If you use the spring parameters defined above and assume a state of equilibrium, the following equations can be written for the state of each endpoint:

$$F_1 - X_1K_1 + (X_2 - X_1)K_2 = 0$$
$$F_2 - (X_2 - X_1)K_2 = 0$$

Therefore,

$$F_1 = (K_1 + K_2)X_1 + (-K_2)X_2$$
$$F_2 = (-K_2)X_1 + K_2X_2$$

If the set of equation is written in matrix form, it will be represented as follows:

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \begin{bmatrix} K_1 + K_2 & -K_2 & X_1 \\ -K_2 & K_2 & X_2 \end{bmatrix}$$

In the above mathematical model, if the spring constants (K_1 and K_2) are known and the deformed shapes (X_1 and X_2) are defined, then the resulting forces (F_1 and F_2) can be determined. Alternatively, if the spring constants (K_1 and K_2) are known and the forces (F_1 and F_2) are defined, then the resulting deformed shape (X_1 and X_2) can be determined.

Various terminologies that are used in the previous example are discussed next.

Stiffness Matrix

In the previous equation, the following part represents the stiffness matrix (K):

$$\begin{bmatrix} K_1 + K_2 & -K_2 \\ -K_2 & K_2 \end{bmatrix}$$

This matrix is relatively simple because it comprises only one pair of springs, but it turns complex when the number of springs increases.

Degrees of Freedom

Degrees of freedom is defined as the least number of independent cordinates required to define the configuration of a system in space. In the previous example, you are only concerned with the displacement and forces. By making one endpoint fixed, you will restrict all degrees of freedom for that particular node. Which means that, there will be no translational or rotational degrees of freedom for that node. But, there are two nodes still have some degrees of freedom. As these two nodes are allowed to translate along the X axis only, they have 1 degree of freedom each considering that no rotational degree of freedom exist in them. The number of the degrees of freedom on free nodes in a model determines the number of equations required to solve a mathematical model.

Boundary Conditions

The boundary conditions are used to eliminate the unknowns in the system. A set of equations that is solvable is meaningless without the input. In the previous example, the boundary condition $X_0 = 0$, and the input forces are F1 and F2. In either ways, the displacements could have been specified in place of forces as boundary conditions and the mathematical model could have been solved for the forces. In other words, the boundary conditions help you reduce or eliminate the unknowns in the system.

Note
The solutions generated by using FEA are always approximate.

Elements and Element Shapes

Before proceeding further, you must be familiar with the concepts of elements and element shapes, because these are the building blocks of FEM. These concepts are discussed next.

Elements

Element is an entity into which the system under study is divided. An element shape is specified by nodes. The shape (area, length, and volume) of an element depends on the nodes with which it is made. An element (triangular shaped) is shown in Figure 1-3.

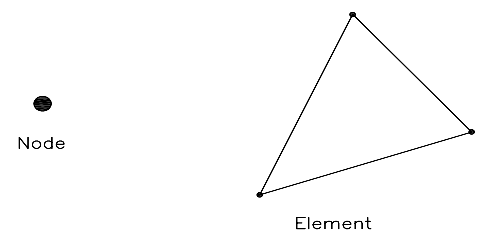


Figure 1-3 A node and an element

Element Shapes

There are many types of element shapes that are further divided into various classes, depending on their uses. The following are some basic element shapes:

Line Element

A line element has the shape of a line or a curve. Therefore, a minimum of two nodes are required to define it. There can be higher order elements that have additional nodes (at the middle of the edge of an element). An element that does not have a node in between its edges is called a linear element. The elements that have nodes in between edges are called quadratic or second order elements. Figure 1-4 shows some line elements.

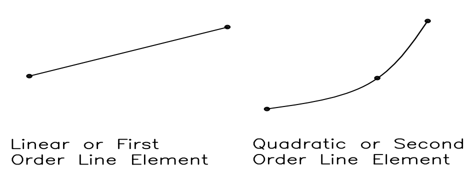


Figure 1-4 Line elements

Area Element

An area element has the shape of a triangle or a quadrilateral; therefore, it requires a minimum of three or four nodes to define it. Some area elements are shown in Figure 1-5.

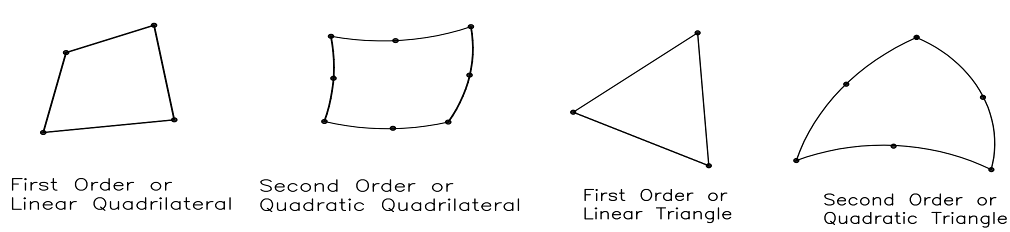


Figure 1-5 The area elements

Volume Element

A volume element has the shape of a hexahedron (8 nodes), wedge (6 nodes), tetrahedron (4 nodes), or a pyramid (5 nodes). Some of the volume elements are shown in Figure 1-6.

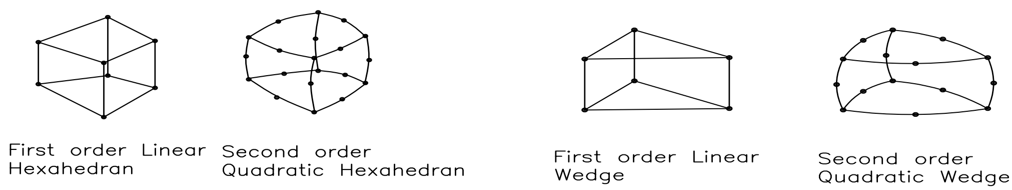


Figure 1-6 *The volume elements*

General Procedure to Conduct Finite Element Analysis

To conduct the finite element analysis, you need to follow certain steps that are given next.

1. Set the type of analysis to be used.
2. Create model.
3. Define the element type.
4. Divide the given geometry into nodes and elements (mesh the model).
5. Apply material properties and boundary conditions.
6. Derive element matrices and equations.
7. Assemble element equations.
8. Solve the unknown parameters at nodes.
9. Interpret the results.

The general process of FEA by using software is divided into three main phases: preprocessing, solution, and postprocessing, refer to Figure 1-7.

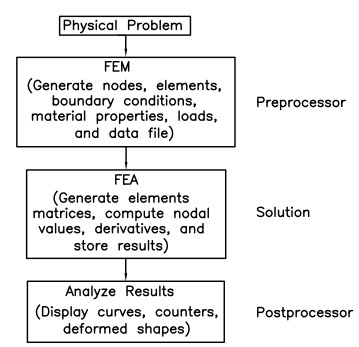


Figure 1-7 *FEA through software*

Preprocessor

The preprocessor is a phase that processes input data to produce output, which is used as input in the subsequent phase (solution). Following are the input data that need to be given to the preprocessor:

1. Type of analysis (structural or thermal, static or dynamic, and linear or nonlinear)
2. Element type
3. Real constants for elements (Cross-sectional area, Moment of Inertia, Shell thickness, and so on)
4. Material properties (Young's Modulus, Poisson's ratio, Spring Constant, Thermal Conductivity, Coefficient of Thermal Expansion, and so on)
5. Geometric model (either created in the FEA software or imported from other CAD packages)

6. FEA model (discretizing the geometric model into small elements)
7. Loading and boundary conditions (defining loads, pressures, moments, temperature, conductivity, convection, constraints (fixed, pinned, or frictionless/symmetrical), and so on.

The input data are preprocessed for the output data and the preprocessor generates the data files automatically with the help of users. These data files are used in the subsequent phase (solution), refer to Figure 1-7.

Solution

The solution phase is completely automatic. The FEA software generates element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used in the subsequent phase (postprocessor) to review and analyze the results through the graphic display and tabular listings, refer to Figure 1-7.

Postprocessor

The output from the solution phase (result data files) is in the numerical form and consists of nodal values of the field variable and its derivatives. For example, in structural analysis, the output of the postprocessor is nodal displacement and stress in elements. The postprocessor processes the result data and displays them in graphical form to check or analyze the result. The graphical output gives the detailed information about the required result data. The postprocessor phase is automatic and generates graphical output in the specified form, refer to Figure 1-7.

FEA SOFTWARE

There are a variety of commercial FEA software packages available in the market. Every CAE software provides various modules for various analysis requirements. Depending on your requirement, you can select a required module for your analysis. Some firms use one or more CAE software and others develop customized version of commercial software to meet their requirements.

Advantages and Limitations of FEA Software

Following are some of the advantages and limitations of FEA software:

Advantages

1. It reduces the amount of prototype testing, thereby saving the cost and time.
2. It gives the graphical representation of the result of analysis.
3. The finite element modeling and analysis are performed in the preprocessor and solution phases, which if done manually would consume a lot of time and in some cases, might be impossible to perform.
4. Variables such as stress and temperature can be measured at any desired point of the model.
5. It helps optimize a design.
6. It is used to simulate the designs that are not suitable for prototype testing.
7. It helps you create more reliable, high quality, and competitive designs.

Limitations

1. It does not provide exact solutions.
2. FEA packages are costly.
3. An inexperienced user can deliver incorrect answers, upon which expensive decisions will be based
4. Results give solutions but not remedies.
5. Features such as bolts, welded joints, and so on cannot be accommodated to a model. This may lead to approximation and errors in the result.
6. For more accurate results, more hard disk space, RAM, and time are required.

KEY ASSUMPTIONS IN FEA

There are four types of key assumptions that must be considered while performing the finite element analysis. These assumptions are not comprehensive, but cover a wide variety of situations applicable to the problem. Moreover, by no means do all the following assumptions apply to all situations. Therefore, you need to consider only those assumptions that are applicable for your analysis problem.

Assumptions Related to Geometry

1. Displacement values will be small so that a linear solution is valid.
2. Stress behavior outside the area of interest is not important. Therefore, geometric simplifications in those areas do not affect the outcome.
3. Only internal fillets in the area of interest will be included in the solution.
4. Local behavior at the corners, joints, and intersection of geometries is of primary interest, therefore, no special modeling of these areas is required.
5. Decorative external features will be assumed insignificant for the stiffness and performance of the part and these external features will be omitted from the model.
6. Variation in the mass due to suppressed features is negligible.

Assumptions Related to Material Properties

1. Material properties will remain in the linear region and the nonlinear behavior of the material property cannot be accepted.
2. Material properties are not affected by the load rate.
3. The component is free from surface imperfections that can produce stress concentration.
4. All simulations will assume room temperature, unless otherwise specified.
5. The effects of relative humidity or water absorption on the material used will be neglected.
6. No compensation will be made to account for the effect of chemicals, corrosives, wears, or other factors that may have an impact on the long term structural integrity.

Assumptions Related to Boundary Conditions

1. Displacements will be small so that the magnitude, orientation, and distribution of the load remains constant throughout the process of deformation.
2. Frictional loss in the system is considered to be negligible.
3. All interfacing components will be assumed rigid.

4. The portion of the structure being studied is assumed as a separate part from the rest of the system, so that any reaction or input from adjacent features is neglected.

Assumptions Related to Fasteners

1. Residual stresses due to fabrication, pre loading on bolts, welding, or other manufacturing or assembly processes will be neglected.
2. All welds between components will be considered as ideal and continuous.
3. The failure of fasteners will not be considered.
4. The load on the threaded portion of the part is supposed to be evenly distributed among the engaged threads.
5. The stiffness of bearings, both in radial and in axial directions, will be considered as infinite or rigid.

TYPES OF ENGINEERING ANALYSES

You can perform different types of analyses using FEA software and these are discusses next.

Structural Analysis

In structural analysis, first the nodal degrees of freedom (displacement) are calculated and then the stress, strains, and reaction forces are calculated from nodal displacements. The classification of structural analysis is shown in Figure 1-8.

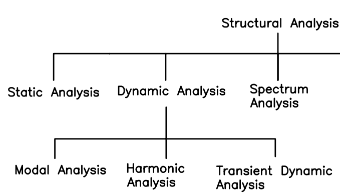


Figure 1-8 *Types of structural analysis*

Static Analysis

In static analysis, the load or field conditions do not vary with respect to time, and therefore, it is assumed that the load or field conditions are applied gradually, not suddenly. The system under this analysis can be linear or nonlinear. The inertia and damping effects are ignored in structural analysis. In structural analysis, the following matrices are solved:

$$[K] \times [X] = [F]$$

Where,

K = Stiffness Matrix
X = Displacement Matrix
F = Load Matrix

The above equation is called the force balance equation for the linear system. If the elements of matrix [K] are the function of [X], the system is known as the nonlinear system. Nonlinear systems include large deformation, plasticity, creep, and so on. The loadings that can be applied in a static analysis include:

1. Externally applied forces and pressures
2. Steady-state inertial forces (such as gravity or rotational velocity)
3. Imposed (non-zero) displacements
4. Temperatures (for thermal strain)
5. Fluences (for nuclear swelling)

The outputs that can be expected from the FEA software are given next.

1. Displacements
2. Strains
3. Stresses
4. Reaction forces

Dynamic Analysis

In dynamic analysis, the load or field conditions vary with the time and are applied suddenly. The system can be linear or nonlinear. The dynamic load includes oscillating loads, impacts, collisions, and random loads. The dynamic analysis is classified into the following three main categories:

Modal Analysis

It is used to calculate the natural frequency and mode shape of a structure.

Harmonic Analysis

It is used to calculate the response of a structure to harmonically time varying loads.

Transient Dynamic Analysis

It is used to calculate the response of a structure to arbitrary time varying loads.

In dynamic analysis, the following matrices are solved:

For the system without any external load:

[M] x Double Derivative of [X] + [K] x [X]= 0

Where,

M = Mass Matrix
K = Stiffness Matrix
X = Displacement Matrix

For the system with external load:

[M] x Double Derivative of [X] + [K] x [X]= [F]

Where,

K = Stiffness Matrix

X = Displacement Matrix

F = Load Matrix

The above equations are called the force balance equations for a dynamic system. By solving the above set of equations, you can extract the natural frequencies of a system. The load types applied in a dynamic analysis are the same as that in a static analysis. The outputs that can be expected from a software are Natural frequencies, Mode shapes, Displacements, Strains, Stresses, and Reaction forces. All these outputs can also be obtained with respect to time.

Spectrum Analysis
This is an extension of the modal analysis and is used to calculate stress and strain due to the response of the spectrum (random vibrations). For example, you can use it to analyze how well a structure will perform and survive in an earthquake.

Buckling Analysis
This type of analysis is used to calculate the buckling load and the buckling mode shape. Slender structures (that is thin and long structures) when loaded in the axial direction, buckle under relatively small loads. For such structures, the buckling load becomes a critical design factor.

Explicit Dynamic Analysis
This type of structural analysis is available only in the ANSYS LS-Dyna program and is used to get fast solutions for large deformation dynamics and complex contact problems, for example, explosions, aircraft crash worthiness, and so on.

Thermal Analysis
The thermal analysis is used to determine the temperature distribution and related thermal quantities such as: Thermal distribution, Amount of heat loss or gain, Thermal gradients, and Thermal fluxes.

All primary heat transfer modes such as conduction, convection, and radiation can be simulated. You can perform two types of thermal analysis, steady-state and transient.

Steady State Thermal Analysis
In this analysis, the system is studied under steady thermal loads with respect to time.

Transient Thermal Analysis
In this analysis, the system is studied under varying thermal loads with respect to time.

Fluid Flow Analysis
This analysis is used to determine the flow distribution and temperature of a fluid. The ANSYS/FLOWTRAN program is used to simulate the laminar and turbulent flow, compressible and electronic packaging, automotive design, and so on. The outputs that can be expected from the fluid flow analysis are Velocities, Pressures, Temperatures, and Film coefficients

Electromagnetic Field Analysis

This type of analysis is conducted to determine the magnetic fields in electromagnetic devices. The types of electromagnetic analyses are Static analysis, Harmonic analysis, and Transient analysis

Coupled Field Analysis

This type of analysis considers the mutual interaction between two or more fields. It is impossible to solve fields separately because they are interdependent. Therefore, you need a program that can solve both the problems by combining them.

For example, if a component is exposed to heat, you may first require to study the thermal characteristics of the component and then the effect of the thermal heating on the structural stability.

Alternatively, if a component is bent in different shapes using one of the metal forming processes and then subjected to heating, the thermal characteristics of the component will depend on the new shape of the component. Therefore, first the shape of the component has to be predicted through structural simulations. This is called as the coupled field analysis.

IMPORTANT TERMS AND DEFINITIONS

Some of the important terms and definitions used in FEA are discussed next.

Strength

When a material is subjected to an external load, the system undergoes a deformation. The material, in turn, offers resistance against this deformation. This resistance is offered by virtue of the strength of the material.

Load

The external force acting on a body is called load.

Stress

The force of resistance offered by a body against the deformation is called stress. The stress is induced in the body while the load is being applied on the body. The stress is calculated as load per unit area.

$$p = F/A$$

Where,

p = Stress in N/mm^2

F = Applied Force in Newton

A = Cross-Sectional Area in mm^2

The material can undergo various types of stresses, which are discussed next.

Tensile Stress
If the resistance offered by a body is against the increase in the length, the body is said to be under tensile stress.

Compressive Stress
If the resistance offered by a body is against the decrease in the length, the body is said to be under compressive stress. Compressive stress is just the reverse of tensile stress.

Shear Stress
The shear stress exists when two materials tend to slide across each other in any typical plane of shear on the application of force parallel to that plane.

Shear Stress = Shear resistance (R) / Shear area (A)

Strain
When a body is subjected to a load (force), its length changes. The ratio of change in the length of the body to its original length is called strain. If the body returns to its original shape on removing the load, the strain is called elastic strain. If the body remains distorted after removing the load, the strain is called plastic strain. The strain can be of three types, tensile, compressive, and shear strain.

Strain (e) = Change in Length (*dl*) / Original Length (*l*)

Elastic Limit
The maximum stress that can be applied to a material without producing the permanent deformation is known as the elastic limit of the material. If the stress is within the elastic limit, the material returns to its original shape and dimension on removing the external stress.

Hooke's Law
It states that the stress is directly proportional to the strain within the elastic limit.

Stress / Strain = Constant (within the elastic limit)

Young's Modulus or Modulus of Elasticity
In case of axial loading, the ratio of intensity of the tensile or compressive stress to the corresponding strain is constant. This ratio is called Young's modulus, and is denoted by *E*.

E = p/e

Shear Modulus or Modulus of Rigidity
In case of shear loading, the ratio of shear stress to the corresponding shear strain is constant. This ratio is called Shear modulus, and it is denoted by C, N, or G.

Ultimate Strength
The maximum stress that a material withstands when subjected to an applied load is called its ultimate strength.

Factor of Safety

The ratio of the ultimate strength to the estimated maximum stress in ordinary use (design stress) is known as factor of safety. It is necessary that the design stress is well below the elastic limit, and to achieve this condition, the ultimate stress should be divided by a 'factor of safety'.

Lateral Strain

If a cylindrical rod is subjected to an axial tensile load, the length (*l*) of the rod will increase (*dl*) and the diameter (Ø) of the rod will decrease (*d*Ø). In short, the longitudinal stress will not only produce a strain in its own direction, but will also produce a lateral strain. The ratio *dl/l* is called the longitudinal strain or the linear strain, and the ratio *d*Ø/Ø is called the lateral strain.

Poisson's Ratio

The ratio of the lateral strain to the longitudinal strain is constant within the elastic limit. This ratio is called the Poisson's ratio and is denoted by *1/m*. For most of the metals, the value of the '*m*' lies between 3 and 4.

Poisson's ratio = Lateral Strain / Longitudinal Strain = 1/m

Bulk Modulus

If a body is subjected to equal stresses along the three mutually perpendicular directions, the ratio of the direct stresses to the corresponding volumetric strain is found to be constant for a given material, when the deformation is within a certain limit. This ratio is called the bulk modulus and is denoted by K.

Stress Concentration

The value of stress changes abruptly in the regions where the cross-section or profile of a structural member changes abruptly. The phenomenon of this abrupt change in stress is known as stress concentration and the region of the structural member that is affected by stress concentration is known as the region of stress concentration. The region of stress concentration needs to be meshed densely to get accurate results.

Bending

When a non-axial force is applied on a structural member, the structural member starts deforming. This phenomenon is known as bending. In case of bending, strains vary linearly from the centerline of a beam to the circumference. In case of pure bending, the value of strain is zero at the centerline. The plane section of the beam is assumed to remain plain even after the bending.

Bending Stress

When a non-axial force is applied on a structural member, some compressive and tensile stresses are developed in the member. These stresses are known as bending stresses.

Creep
At elevated temperature and constant load, many materials continue to deform but at a slow rate. This behavior of materials is called creep. At a constant stress and temperature, the rate of creep is approximately constant for a long period of time. After a certain amount of deformation, the rate of creep increases, thereby causing fracture in the material. The rate of creep depends highly on both the stress and the temperature.

Classification of Materials
Materials are classified into three main categories: elastic, plastic, and rigid. In case of elastic materials, the deformation disappears on the removal of load. In plastic materials, the deformation is permanent. A rigid material does not undergo deformation when subjected to an external load. However, in actual practice, no material is perfectly elastic, plastic, or rigid. The structural members are designed such that they remain in the elastic conditions under the action of working loads. All engineering materials are grouped into three categories that are discussed next.

Isotropic Material
In case of Isotropic materials, material properties do not vary with direction, which means they have the same material properties in all directions. Material properties are defined by Young's modulus and Poisson's ratio.

Orthotropic Material
In case of orthotropic materials, material properties vary with direction and are specified in three orthogonal directions. Such materials have three mutually perpendicular planes of material symmetry. Material properties are defined by separate Young's modulus and Poisson's ratios along each axis.

Anisotropic Material
In case of Anisotropic materials, material properties vary with direction, but there is no plane of material symmetry. This means they do not behave in the same way in all directions.

Aspect Ratio
Aspect ratio is defined as the ratio of the longest side to the smallest side of an element.

Axisymmetry
Model that can be defined by rotating its cross-section by 360-degrees about an axis is known as axisymmetry model.

Degrees of Freedom (DOF)
Degrees of freedom is defined as the freedom of a given point to move in any direction in space.

There are six DOFs for any point in 3-dimensional (3D) space:
3 translational DOFs (one each in the X,Y, and Z directions) and
3 rotational DOFs (one each about the X, Y, and Z axes).

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. FEA simulates the loading conditions of a model and determines its response under those conditions. (T/F)
2. A linear line element has a maximum of two nodes. (T/F)
3. A quadratic line element has a node in the middle. (T/F)
4. An area element should always be triangular in shape. (T/F)
5. You cannot import an external geometry file into an FEA software. (T/F)
6. The nodes define the shape of an element. (T/F)
7. A minimum of ____________ nodes are required to define a line element.
8. A minimum of ____________ nodes are required to define an area element.
9. A minimum of ____________ nodes are required to define a volume element.

Review Questions

Answer the following questions:

1. The Finite Element Method gives exact solutions to problems. (T/F)
2. In FEM, the geometry is discretized into small parts, known as elements. (T/F)
3. In space, a rigid body has six degrees of system. (T/F)
4. In dynamic analysis, the boundary conditions are a function of __________.
5. Modal analysis is used to calculate the _______________ frequencies of a model.
6. Hooke's law states that stress is directly proportional to ____________ within elastic limit.

Answers to Self-Evaluation Test

1. T, **2.** T, **3.** T, **4.** F, **5.** F, **6.** T, **7.** two, **8.** three, **9.** four