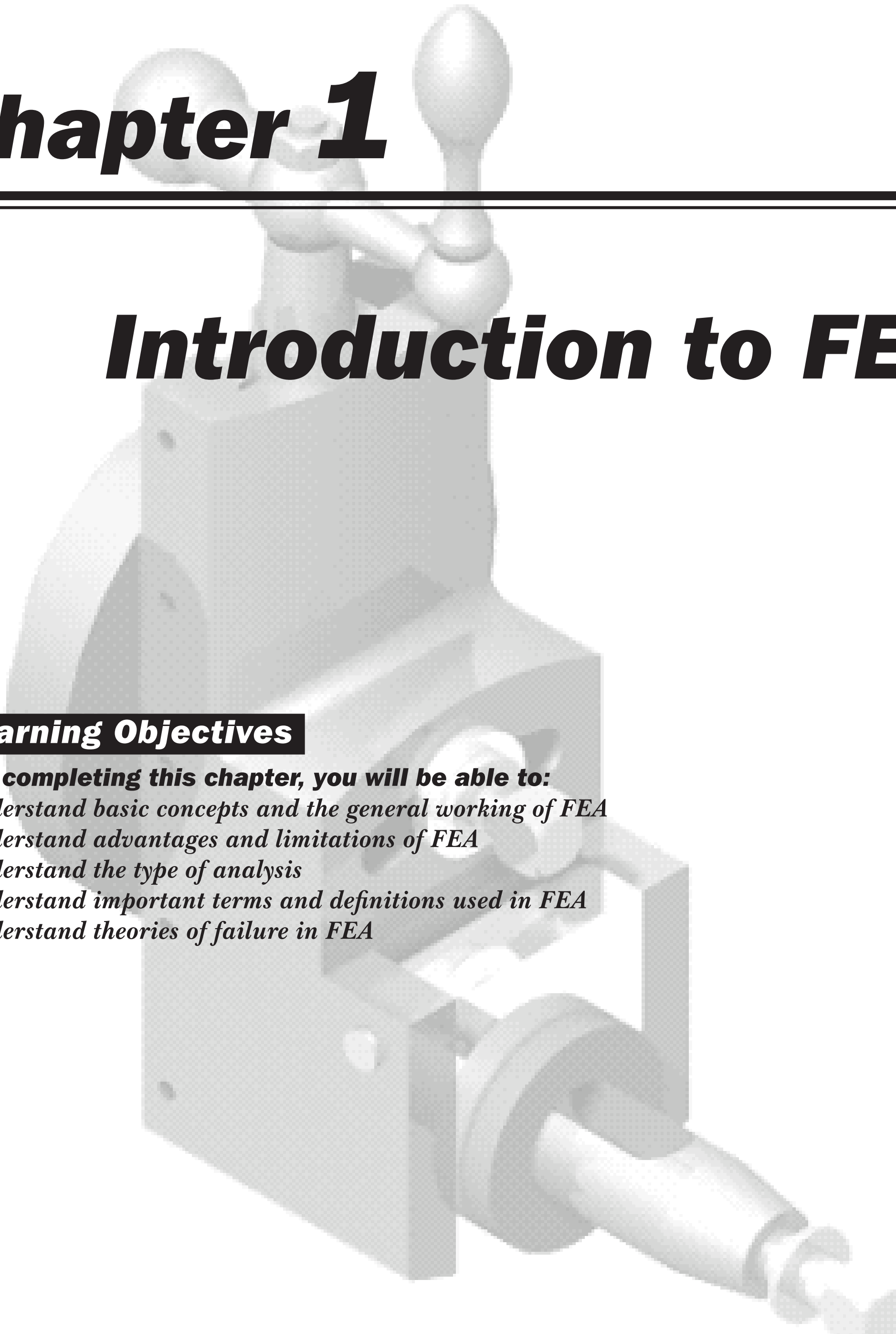
Chapter 1

Introduction to FEA



Learning Objectives

After completing this chapter, you will be able to:

- *Understand basic concepts and the general working of FEA*
- *Understand advantages and limitations of FEA*
- *Understand the type of analysis*
- *Understand important terms and definitions used in FEA*
- *Understand theories of failure in FEA*

INTRODUCTION TO FEA

The finite element analysis (FEA) is a computing technique that is used to obtain approximate solutions to the boundary value problems in engineering. In this analysis, a numerical technique called the finite element method (FEM) is used to solve boundary value problems. FEA involves a computer model of a design that is loaded and analyzed for specific results. The finite element analysis was first introduced by Richard Courant in 1943. He used the Ritz method of numerical analysis and minimization of variational calculus for getting approximate solutions to vibration systems. Later, the academic and industrial researchers created the finite element method for structural analysis.

The concept of FEA can be explained through a simple example of measuring 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 endpoint of each of these segments by a straight line. Now, you can easily measure the length of the straight line, and thus the perimeter of the circle.

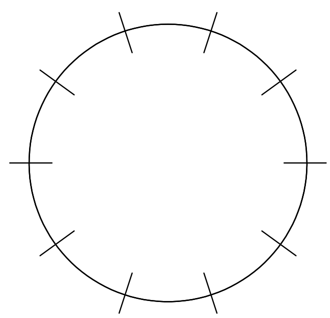


Figure 1-1 *The circle divided into equal segments*

If the number of segments into which the circle is divided is less, as shown in Figure 1-1, you will not get accurate results. For accuracy, divide the circle into more number of segments. However, with more segments, the effort required is more. The same concept applies to FEA also, and therefore, there is always a compromise between accuracy and speed while using this method, which 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 crucial. Nowadays, the simplest of the products rely on the FEA for their design evaluation.

The FEA simulates the loading conditions of a design and determines the design response in those conditions. The design is modeled using the discrete building blocks called elements. Each element has some equations that describe its response to certain loads. The sum of the responses of all the elements in a model gives the total response of the design.

General Working of FEA

Better knowledge of FEA will help you build more accurate models. It will also help you understand the backend working of Autodesk Simulation Mechanical. A simple model is discussed here to give you a brief overview of FEA.

Figure 1-2 shows a spring assembly that represents a simple two-spring element model. These two springs are connected in series and one of the springs is fixed at the left endpoint, refer to Figure 1-2. The stiffness of the springs is represented by spring constants K_1 and K_2. The endpoints of each spring is restricted to the displacement or the translation in 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 forces acting on each endpoint of the springs are represented by F_1 and F_2.

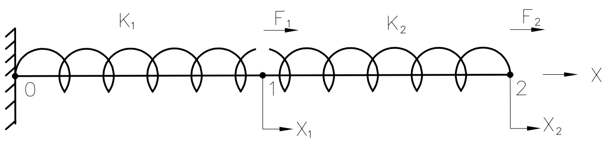


Figure 1-2 *Representation of a two-spring assembly*

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

$$F = KX$$

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 above set of equation is written in matrix form, it will be modified as follows:

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

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

This type of spring system may be complicated to define, but it involves most of the key terminologies used in FEA. These FEA terminologies are discussed next.

1. Stiffness Matrix
2. Degree of Freedom
3. Boundary Conditions

Stiffness Matrix

The stiffness matrix represents the resistance offered by a body to withstand the load applied. In the previous equation, the following part represents the stiffness matrix (K):

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

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

Degree of Freedom

The Degree of freedom is defined as the ability of a node to translate or transmit the load. In the previous example, you are only concerned with the displacement and forces. By making one endpoint fixed, one degree of freedom for displacement is removed. So, now the model has two degrees of freedom. The number of degrees of freedom in a model determines the number of equations required to solve the mathematical model.

Boundary Conditions

The boundary conditions are used to eliminate the unknowns in a system. A set of equations that is solvable is meaningless without the input. In the previous example, the boundary condition was $X_0 = 0$, and the input forces were 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 unknowns in the system.

The FEA technique needs the finite element model (FEM) for its final solution as it does not use the solid model. FEM consists of nodes, keypoints, elements, material properties, loading, and boundary conditions.

Nodes, Elements, and Element Types

Before proceeding further, you must be familiar with commonly used terms such as nodes, elements, and element types. These terms are discussed next.

Nodes

An independent entity in space is called a node. Nodes are similar to the points in geometry and represent the corner points of an element. You can change the shape of an element by moving the nodes in space. The shape of a node is shown in Figure 1-3.

Elements
An element is an entity into which the system under study is divided. The shape (area, length, and volume) of an element is specified by nodes. Figure 1-3 shows a triangular shaped element.

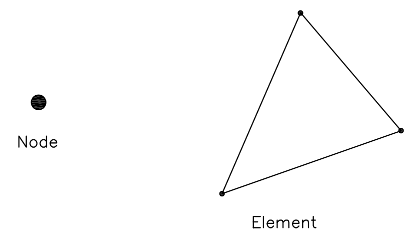


Figure 1-3 *A node and an element*

Element Types
The following are the basic types of the elements:

Point Element
A point element is in the form of a point and therefore has only one node.

1D Element
A 1D element has the shape of a line or 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 the element). The element that does not have a node at the middle of the edge of the element is called a linear element. The elements with node at the mid of the edges are called quadratic or second order elements. Figure 1-4 shows some line elements.

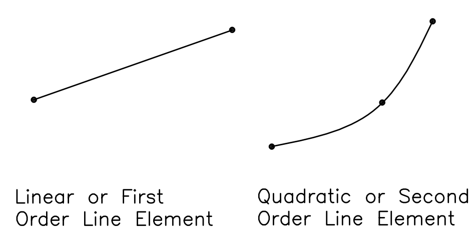


Figure 1-4 *The 1D elements*

2D Element
An 2D element has the shape of a quadrilateral or a triangle, therefore it requires a minimum of three or four nodes to define it. Some of the 2D elements are shown in Figure 1-5.

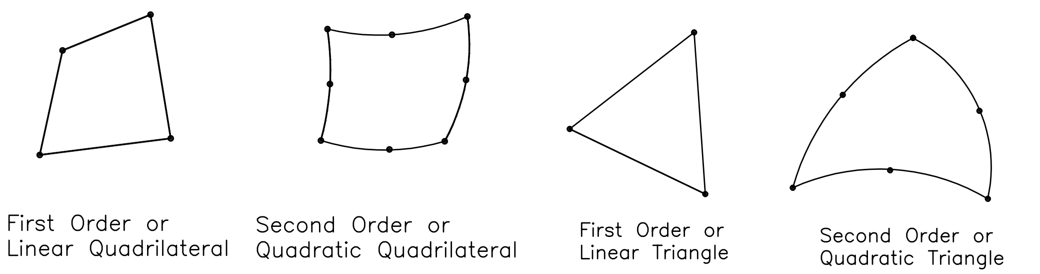


***Figure 1-5** The 2D elements*

Note
In this chapter, only the basic introduction of element types has been covered.

3D Element

A 3D element has the shape of a hexahedron (8 nodes), wedge (6 nodes), tetrahedron (4 nodes), or a pyramid (5 nodes). Some of the 3D elements are shown in Figure 1-6.

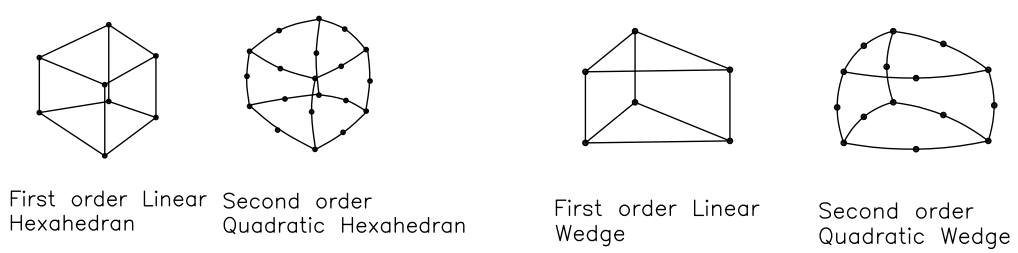


***Figure 1-6** The 3D elements*

Areas for Application of FEA

FEA is a very important tool for designing. It is used in the following areas:

1. Structural strength design
2. Structural interaction with fluid flows
3. Shock analysis
4. Acoustics
5. Thermal analysis
6. Vibrations
7. Crash simulations
8. Fluid flows
9. Electrical analysis
10. Mass diffusion

11. Buckling problems
12. Dynamic analysis
13. Electromagnetic analysis
14. Coupled analysis, and so on.

General Procedure of Conducting Finite Element Analysis

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

1. Set the type of analysis to be carried out.
2. Create or import the model.
3. Define the element type.
4. Divide the given problem into nodes and elements (generate a mesh).
5. Apply material properties and boundary conditions.
6. Solve the unknown quantities at nodes.
7. Interpret the results.

FEA through Autodesk Simulation Mechanical

In Autodesk Simulation Mechanical, the general process of finite element analysis is divided into three main phases, namely preprocessor, solution, and postprocessor, refer to Figure 1-7.

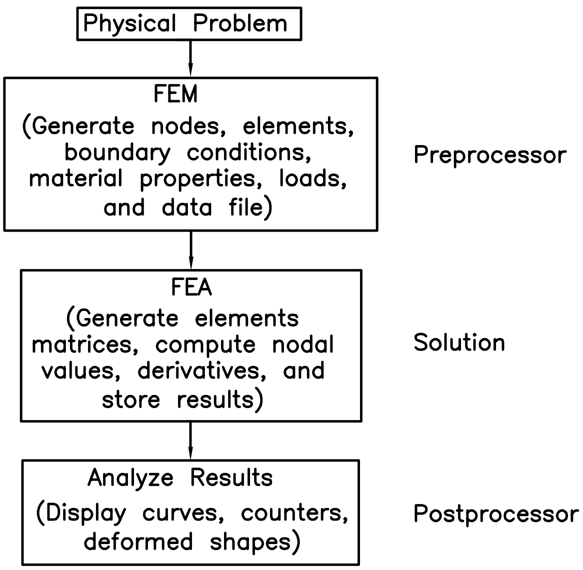


Figure 1-7 FEA through Autodesk Simulation Mechanical

Preprocessor

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

1. Analysis type (structural or thermal, static or dynamic, and linear or nonlinear)
2. Element type
3. Real constants
4. Material properties
5. Geometric model
6. Meshed model
7. Loadings and boundary conditions

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

Solution

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

Postprocessor

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

Effective Utilization of FEA

Some prerequisites for effective utilization of FEA from the perspective of engineers and FEA software are discussed next.

Engineers

An engineer who wants to work with this tool must have sound knowledge of Strength of Materials (for structural analysis), Heat Transfer, Thermodynamics (for thermal analysis), and a good analytical/designing skill. Besides this, the engineer must have a fair knowledge of advantages and limitations of the FEA software being used.

Software

The FEA software should be selected based on the following considerations:

1. Analysis type to be performed.
2. Flexibility and accuracy of the tool.
3. Hardware configuration of your system.

Nowadays, the CAE / FEA software can simulate the performance of most of the systems. In other words, anything that can be converted into a mathematical equation can be simulated using the FEA techniques. Usually, the most popular principle of GIGO (Garbage In Garbage Out) applies to FEA. Therefore, you should be very careful while giving/accepting the inputs for analysis. The careful planning is the key to a successful analysis.

Advantages and Limitations of FEA Software

Following are the advantages and limitations of FEA software:

Advantages

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

Limitations

1. It provides approximate solutions.
2. FEA packages are costly.
3. Qualified personnel are required to perform the analysis.
4. The results give solutions but not remedies.
5. Features such as bolts, welded joints, and so on cannot be accommodated to the model. This may lead to approximation and errors in the result obtained.
6. For more accurate result, more computer space and time are required.

KEY ASSUMPTIONS IN FEA

There are four basic assumptions that affect the quality of the solution and must be considered before carrying out finite element analysis. These assumptions are not comprehensive, but cover a wide variety of situations applicable to the problem. Make sure to use only those assumptions that apply to the analysis under consideration.

Assumptions Related to Geometry

1. When the displacement is small, the linear solution can be consider.
2. Stress behavior outside the area of interest is not important so the geometric simplifications in those areas will 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 will be omitted from the model.
6. The variation in mass due to the suppressed features is negligible.

Assumptions Related to Material Properties

1. Material properties will remain in the linear region and nonlinear behavior of the material property cannot be accepted. For example, it is understood that either the stress levels exceeding the yield point or the excessive displacement will cause a component failure.
2. Material properties are not affected by the load rate.
3. The component is free from surface imperfections that can produce stress risers.
4. All simulations will assume room temperature unless specified otherwise.
5. The effect 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 a part separate from the rest of the system. As a result, the reaction or input from the adjacent features is neglected.

Assumptions Related to Fasteners

1. Residual stresses due to fabrication, preloading on bolts, welding, or other manufacturing or assembly processes will be neglected.
2. All the welds between the components will be considered ideal and continuous.
3. The failure of fasteners will not be considered.
4. Loads on the threaded portion of the parts is supposed to be evenly distributed among the engaged threads.
5. Stiffness of bearings, radially or axially, will be considered infinite or rigid.

TYPES OF ANALYSIS

The following types of analysis can be performed using FEA software:

1. Structural analysis
2. Thermal analysis

3. Fluid flow analysis
4. Electromagnetic field analysis
5. Coupled field analysis

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 the nodal displacements. The classification of the structural analysis is shown in Figure 1-8.

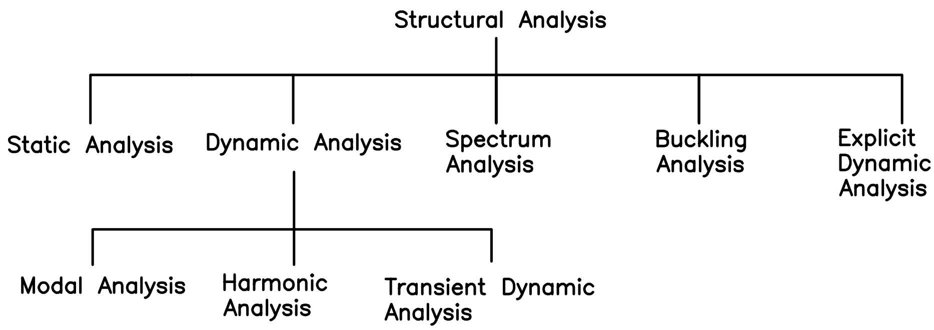


Figure 1-8 *Types of structural analysis*

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 analysis can be linear or nonlinear. 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 as the force balance equation for a linear system. If the elements of matrix [K] are a function of [X], the system is known as a 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 a FEA software are given next.

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

Dynamic Analysis

In dynamic analysis, the load or field conditions do vary with time. The assumption here is that the load or field conditions 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 the structure to harmonically time varying loads.

Transient Dynamic Analysis

It is used to calculate the response of the 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 as force balance equations for a dynamic system. By solving the above set of equations, you will be able to extract the natural frequencies of a system. The load types applied in the dynamic analysis are the same as that in the static analysis. The outputs that can be expected from a software are:

1. Natural frequencies
2. Mode shapes

3. Displacements
4. Strains
5. Stresses
6. Reaction forces

All the outputs mentioned above can be obtained with respect to time.

Spectrum Analysis
This is an extension of the modal analysis and is used to calculate the stress and strain due to the response of the spectrum (random vibrations).

Buckling Analysis
This type of analysis is used to calculate the buckling load and the buckling mode shape. The slender structures and the structures with slender part 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 used to calculate fast solutions for large deformation dynamics and complex contact problems.

Thermal Analysis
Thermal analysis is used to determine the temperature distribution and related thermal quantities such as:

1. Thermal distribution
2. Amount of heat loss or gain
3. Thermal gradients
4. Thermal fluxes

All the 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. It 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:

1. Velocities
2. Pressures
3. Temperatures
4. Film coefficients

Electromagnetic Field Analysis

This type of analysis is used to determine the magnetic fields in electromagnetic devices. The types of electromagnetic analyses are:

1. Static analysis
2. Harmonic analysis
3. Transient analysis

Coupled Field Analysis

This type of analysis considers the mutual interaction between two or more fields. It is impossible to solve the fields separately because they are interdependent. Therefore, you need a program that can solve both the physical 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 into 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 and therefore the shape of the component has to be predicted through structural simulations first. This is called as 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 deformation occurs and the resistance against this deformation offers by the material is by the virtue of its strength.

Load

The external force acting on a body is called the 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 applied on it. 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 length, the body is said to be under tensile stress.

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

Shear Stress
Shear stress exists when two materials tend to slide over each other in opposite direction. Note that in any typical plane of shear, the force parallel to the plane is usually known as shear stress.

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

Strain
When a body is subjected to a load (force), its length changes. The ratio of the change in length to the original length of the member is called strain. If the body returns to its original shape on removing the load, the strain is called as elastic strain. If the metal remains distorted, the strain is called as plastic strain. The strain can be of three types, namely tensile, compressive, or 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 permanent deformation is known as the elastic limit of the material. If the stress applied is within the elastic limit and on the removal of the stress, the material will return to its original shape and dimension.

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 tensile or compressive stress to the corresponding strain is constant. This ratio is called Young's modulus, and it is denoted by E.

E = p/e

Where,
p = Stress
e = strain

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 can withstand when load is applied is called its ultimate strength.

Factor of Safety

The ratio of the ultimate strength to the estimated maximum stress (design stress) is known as factor of safety. It is necessary that the design stress should be 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 (ϕ) of the rod will decrease ($d\phi$). 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 linear strain, and the ratio $d\phi/\phi$ is called the lateral strain.

Poisson's Ratio

The ratio of lateral strain to the longitudinal strain is constant within the elastic limit. This ratio is called as 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 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.

Creep
At elevated temperatures and constant stress or 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 this period and after a certain amount of deformation, the rate of creep increases thereby causing fracture in the material. The rate of creep is highly dependent 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 any 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 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, the material properties do not vary with direction, which means that they have same material properties in all directions. The material properties are defined by Young's modulus and Poisson's ratio.

Orthotropic Material
In case of Orthotropic material, the material properties vary with the change in direction. They have three mutually perpendicular planes of material symmetry. The material properties are defined by three separate Young's modulus and Poisson's ratios.

Anisotropic Material
In case of Anisotropic material, the material properties vary with the change in direction, but in this case, there is no plane of material symmetry.

THEORIES OF FAILURE
The following are the theories of failure.

Von Mises Stress Failure Criteria
The von Mises stress criterion is also called Maximum Distortion Energy theory. The theory states that a ductile material starts yielding at a location when the von Mises stress becomes equal to the stress limit. In most cases, the yield strength is used as the stress limit.

Maximum Shear Stress Failure Criterion

The Maximum Shear Stress failure criterion is based on the Maximum Shear Stress theory. This theory predicts failure of a material when the absolute maximum shear stress reaches the stress limit that causes the material to yield in a simple tension test. The Maximum Shear Stress criterion is used for ductile materials.

Maximum Normal Stress Failure Criteria

This criterion is used for brittle materials. It assumes that the ultimate strength of the material in tension and compression is the same. This assumption is not valid in all the cases. For example, cracks considerably decreases the strength of the material in tension while their effect is not significant in compression because the cracks tend to close. Brittle materials do not have a specific yield point and hence it is not recommended to use the yield strength to define the stress limit for this criterion.

SELF-EVALUATION TEST

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

1. The FEA is a computing technique that is used to obtain approximate solutions to the boundary value problems in engineering. (T/F)

2. The Model Analysis is used to calculate the natural frequency and mode shape of a structure. (T/F)

3. The degree of freedom is defined as the ability of a node to translate or transmit the load. (T/F)

4. A 1D element has the shape of a line or curve. (T/F)

5. The __________ are used to eliminate the unknowns in a system.

6. An element shape is specified by __________.

7. In Autodesk Simulation Mechanical, the general process of finite element analysis is divided into three main phases: __________ , __________ , and __________.

8. In __________ analysis, the load or field conditions do not vary with respect to time.

REVIEW QUESTIONS

Answer the following questions:

1. The stiffness matrix represents the resistance offered by a body to withstand the load applied. (T/F)

2. The boundary conditions are used to eliminate the unknowns in a system. (T/F)

3. In Autodesk Simulation Mechanical, the solution phase of analysis is completely automatic. (T/F)

4. When the stiffness of material is the function of displacement then the behavior of material is known as __________.

5. In __________ analysis, the load or field conditions vary with respect to time.

6. The ratio of the change in length to the original length of the member is called __________.

7. The __________ states that the stress is directly proportional to the strain within the elastic limit.

8. A 3D element can have the shape of hexahedron (8 nodes), __________ , __________ , or __________ .

Answers to Self-Evaluation Test

1. T, **2.** T, **3.** T, **4.** T, **5.** boundary conditions, **6.** nodes, **7.** preprocessor, solution, postprocessor, **8.** Static