CONCEPT OF FINITE ELEMENT ANALYSIS

WHAT IS FEA? FEA IN ENGINEERING

Finite Element Analysis is a numerical method for solving problems of engineering and physics fields. It is an extension of matrix methods of structural analysis. It is based on Finite Element Method used to simulate(create a virtual version) and analyze how the systems reacts under various physical conditions like stresses, heat transfer, tempearatures,etc… It breaks down a complex object/structure into smaller and simpler parts which are called as Finite Elements.

The main purpose of FEA is to solve practical engineering problems where exact solutions cannot be find.FEA is invaluable for Engineers to solve complex problems enhancing safety and performance by saving the time and cost. Other useful ways are like design optimization(test&refine), ensures verification and validation, customization for various fields like civil,aerospace,mechanical,etc. It is applicable for structural analysis, thermal analysis, vibration and dynamic analysis, fatigue and failure prediction, fluid-structure interaction. FEA reduces need for physical prototypes, helps in decision-making and planning, and avoids risk by providing nearly accurate values and detailed results.

BASIC STEPS INVOLVED IN FEA

The basic steps involved in any finite element analysis consist of Preprocessing phase, Solution phase, postprocessing phase and iteration phase.

Preprocessing phase

Create and separate the solution domain into finite elements, that is, elements and nodes.[define shape and dimensions of the object to be analyzed in the CAD softaware]

Assume a shape function(approximate continuous function) to represent the physical behaviour of an element.[assign the material properties in the CAD software]

Develop the equations for an element.[apply the mesh in CAD software]

Assemble the elements and construct the global stiffness matrix(the matrix represents the resistance of the element to change when subjected to external influences).

Apply the boundary conditions, initial conditions, and loadings.

Solution phase

Solve a set of linear or non-linear algebraic equations simultaneously to obtain nodal results.

In softwares, the system of equations is assembled and solved using numerical methods (e.g., matrix methods). Solvers compute variables like displacement, stress, strain, temperature, etc., at each node or element.

Postprocessing phase

This phase includes obtaining other data like Visualization, Results Evaluation and Validation.

Iteration phase(optional)

Modify geometry, mesh size, material properties, or boundary conditions based on results.

Rerun simulations to refine and optimize the design.

ELEMENT,NODE AND MESH QUALITY

An **element** is a small, simplified part of the model's geometry used in the simulation. The finite element procedure reduces unknowns to a finite number by dividing the solution region into small parts(elements).

The approximating functions are defined in terms of field variables of specified points(corners or midpoints of elements where calculations) are called **nodes or nodal points**. Elements are connected through nodes.

A **mesh** is the entire network of elements and nodes that represents the model in FEA. It defines how the geometry is divided for analysis.

Mesh quality affects the accuracy, efficiency, and reliability of the FEA results. High-quality meshes produce more precise results, especially in regions of high stress or curvature. A good mesh helps the numerical solution converge properly without errors. A well-optimized mesh balances accuracy and speed—too coarse gives poor results; too fine increases computing time. Poor-quality (like distorted or skewed) elements can cause numerical instability or wrong results. Problems like uneven stress distribution, Slow or failed solver convergence, Misleading results, especially near holes, sharp corners, or interfaces.

BOUNDARY CONDITIONS

The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. They specify constraints or loads at the edges or surfaces of the model to simulate real-world behavior.

Types of Boundary Conditions:

1) Displacement Boundary Conditions: These specify the movement of nodes like how much the distance it can move and also restrict the translation or rotation of certain parts of model by fixing the motion.
   Ex: fixed, pinned, prescribed(defined motion)
2) Force Boundary Conditions: These specify the external loads or pressures applied to the system.
   Ex: point loads, distributed loads, thermal loads
3) Symmetry Boundary Conditions: These are used to reduce problem size without disturbing accuracy of results when the model exhibits symmetry.
4) Contact Boundary Conditions: These are used to simulate interactions between different parts of the model like sliding behaviour, friction coefficients, etc…
5) Thermal Boundary Conditions: These are used to simulate Heat Transfer scenarios which specify Temperature distribution, heat flux, convection, etc…

STIFFNESS MATRIX

Stiffness Matrix is a mathematical matrix which represents the geometrical and material behaviour information that indicates the resistance of an element to deformation when subjected to loading or externally influencing factors. Stiffness Matrix is of two types: global and element.

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

The stiffness matrix is important in FEA to predict how a structure behaves under load and also to solve the FEA problems. It has the ability to handle complex structures, more flexible and has numerical stability.

For Example: Two-Node Linear Spring Element

Consider a spring of stiffness k connected between two nodes, Node 1 and Node 2.The spring resists extension or compression.Displacements at Node 1 and Node 2 are u1 and u2, respectively.

For a 1D spring, the element stiffness matrix is:

$$[Ke]=[k\ -k\ -k\ k]$$

This matrix relates nodal displacements {u} to nodal forces {F}:

$$[k\ -k\ -k\ k][u1\ u2]=[F1\ \ F2]$$

If both nodes are pulled equally, the spring doesn't stretch, so no force is generated.If one node is moved relative to the other, the spring develops force trying to restore the original length.

Let’s assume: Spring stiffness k=100 N/m

Node 1 is fixed: u1=0

Node 2 is displaced by u2=0.01 m

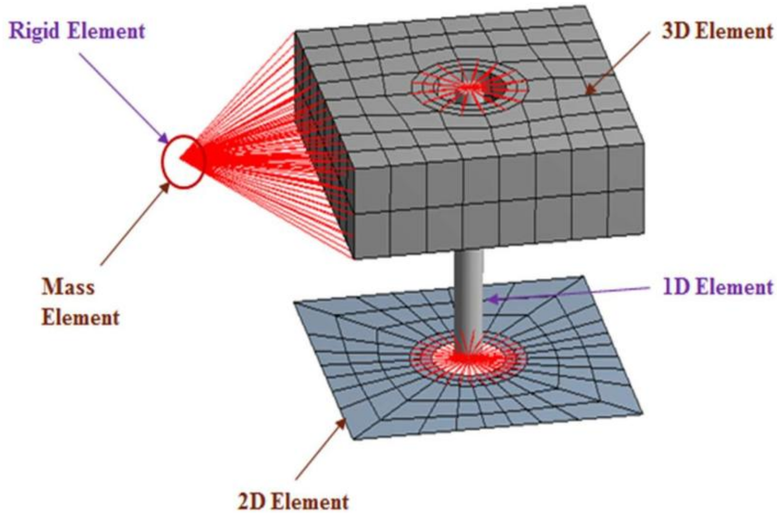
Now calculating the force at each node, we get, [100 −100 −100 100] [0 0.01] = [−1 1] N

Node 1 feels a force of –1 N (pulling left)

Node 2 feels a force of +1 N (pulling right)

1D, 2D AND 3D ELEMENTS

Feature	**1D Element**	**2D Element**	**3D Element**
Geometry	Line	Surface (flat shape)	Solid (volume)
DOFs per node	1–3	2	3
Examples	Beams,Rods,Pipes	Sheet Metal Components, Plastic Panels	Cube, Tetrahedron
Applications	Frames, Bridges	Walls, Thin Plates	Machine Parts, Castings
Simulates	Axial/Bending	In-plane stress/strain	Full stress state



1D elements are the simplest lines between two nodes where the case is one dimension is larger than other. 2D elements are like rectangles where the case is when two dimensions are larger than the third one. In case of 3D there is no single dimension domination, all are comparable like tetrahedrons, cuboids, etc.

SHAPE FUNCTION

Shape function is a mathematical function in FEM used to interpolate(insert) the values of a physical quantity. It provides information about how the value of a field variable varies inside an element, using the values at its nodes.

Simple Example: 1D Linear Bar Element: A bar of length L with two nodes:

Node 1 at $x=0$

Node 2 at $x=L$

In the finite element approach, the nodal values of the field variable are treated as unknown constants that are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes. The interpolation functions are predetermined, known functions of the independent variables; and these functions describe the variation of the field variable within the finite element.

Let the displacements at Node 1 and 2 be $u1$ and $u2$, respectively.

The Linear Shape Functions:

$$N1(x)=1-(x/L), N2(x)=x/L$$

These shape functions satisfy:

At Node 1 ($x = 0$):

$N1=1, N2=0$

At Node 2 ($x = L$):

$N1=0, N2=1$

The displacement at any point xxx along the element is given by:

$$u(x)=N1(x)\cdot u1+N2(x)\cdot u2$$

So, the displacement **smoothly varies** between the two nodes, based on how far along the bar you are.

For the three-node triangle example, the field variable is described by the approximate relation

$$\phi(x, y) = N1(x, y)\ \phi1 + N2(x, y)\ \phi2 + N3(x, y)\ \phi3$$

where $\phi1$, $\phi2$, and $\phi3$ are the values of the field variable at the nodes, and N1, N2, and N3 are the interpolation functions, also known as shape functions or blending functions.

FEA vs CLASSICAL METHOD

FEA provides exact solutions and works best for simple shapes. It assumes uniform and linear materials. It is difficult to apply for larger, multi-body systems. It is limited to only ideal or standard boundary conditions. FEA consumes more time for complex problems. The visualization is limited to graphs and equations only. It is used in case of solving Industrial-level problems. It divides the model into finite elements, simulate and provide detailed results.

Classical or Analytical Methods generally use Beam Theory. It provides approximate solutions, handles complex geometries too. It can also handles non-linear, composite and varying properties too. It supports complex, real-world boundary conditions. It can be applied to large and complex systems. It is highly flexible and also efficient with modern tools. It provides detailed graphical output without any limitation. It is used in case of learning basic theory.

COMMON SOFTWARE TOOLS USED IN FEA

Some of the Commercial options are

- Ansys
- Abaqus
- Solidworks Simulation
- COSMOL Multiphysics
- STAAD.Pro
- GT STRUDL
- ETABS
- MSC Nastran/Patran
- Autodesk Fusion360
- Altair Hyperworks

Some of the Open-source options are

- OpenFOAM
- SimScale
- Code_Aster
- Elmer FEM
- CalculiX
- Opensees
- FreeCAD
- Fenics Project
- Salome-Meca

REAL-WORLD APPLICATIONS

EXAMPLE 1: Automotive Crash Testing

Step 1: Define the problem — simulate a frontal collision.
Step 2: Create a 3D CAD model of the car body and frame.
Step 3: Assign material properties like steel or aluminum behavior under impact.
Step 4: Mesh the model finely in critical areas (bumper, chassis).
Step 5: Apply boundary conditions such as road contact and fixed supports.
Step 6: Apply impact forces to simulate crash speed.
Step 7: Solve using nonlinear dynamic solvers.
Step 8: Post-process to identify deformation and stress zones.
Step 9: Validate with crash test data and refine the design.

EXAMPLE 2: Aerospace Satellite Structure

Step 1: Define goal — analyze satellite stress during launch.
Step 2: Model satellite frame, solar panels, and internal parts in CAD.
Step 3: Assign aerospace-grade material properties like titanium or composites.
Step 4: Generate a mesh, using finer mesh in joints and brackets.
Step 5: Apply constraints representing launch vehicle attachment points.
Step 6: Apply loads from launch acceleration and vibration spectra.
Step 7: Solve using static and modal analysis.
Step 8: Review stress, displacement, and vibration modes.
Step 9: Validate with vibration test results and optimize weight.

EXAMPLE 3: Biomedical Hip Implant Design

Step 1: Define objective — test implant under human body load.
Step 2: Create 3D model of bone-implant assembly.
Step 3: Assign nonlinear material properties for bone and implant.
Step 4: Mesh finely near the bone-implant interface.
Step 5: Apply constraints where bones are naturally supported.
Step 6: Apply joint loading to simulate walking forces.
Step 7: Solve using static structural analysis.
Step 8: Check for stress concentrations and micromotions.
Step 9: Validate with lab testing and refine design.