Brief Information about Finite Element Method (FEM)

Definition of FEM

The finite element method (FEM) is a widely used **numerical technique for solving differential equations in engineering and mathematical modeling.** The method is known for its **accuracy and efficiency in solving complex problems.** It is commonly applied to **structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential.**

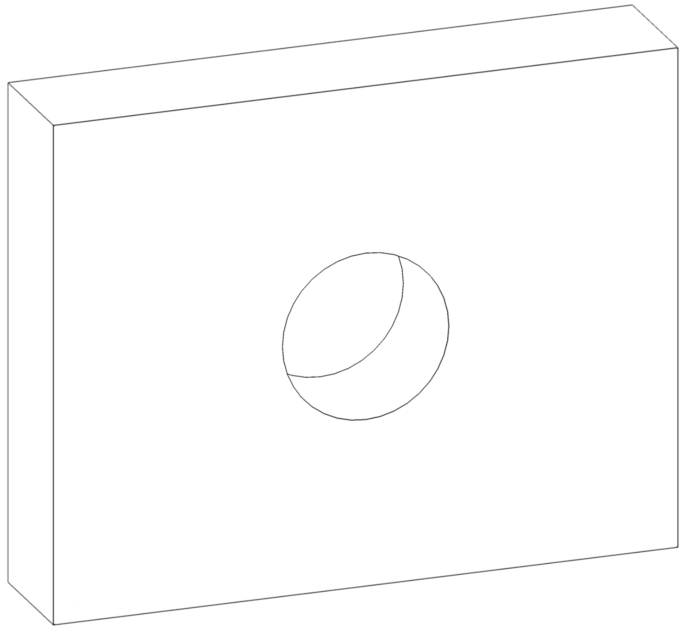
The finite element method (FEM) is based on the idea of **dividing the computational domain into small patches** and finding local solutions that satisfy the differential equation within the boundaries of these patches. **By combining these individual solutions, a global solution can be obtained.**

General Steps of FEM

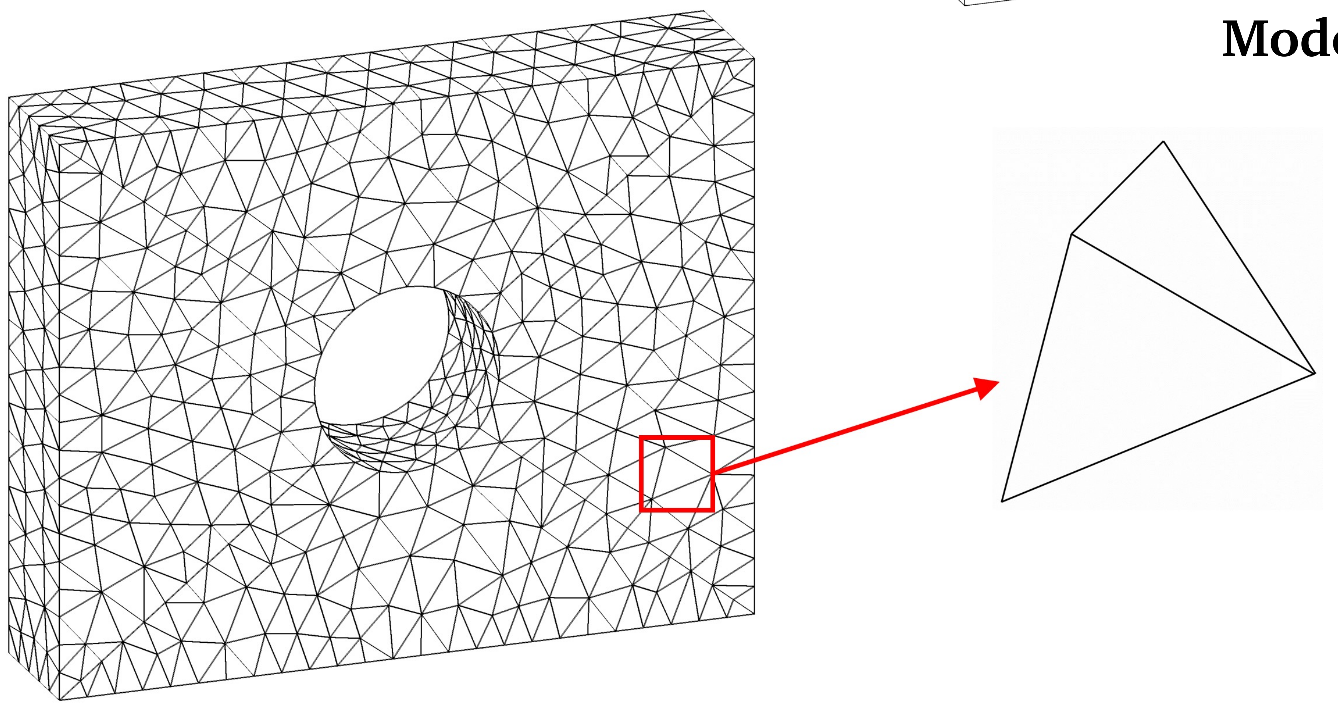
1. Discretize the continuum and select the element types
2. Select interpolation functions
3. Find the element properties
4. Assemble the element equations
5. Solve the global equation system
6. Compute additional results

1. Discretize the Continuum and Select the Element Types

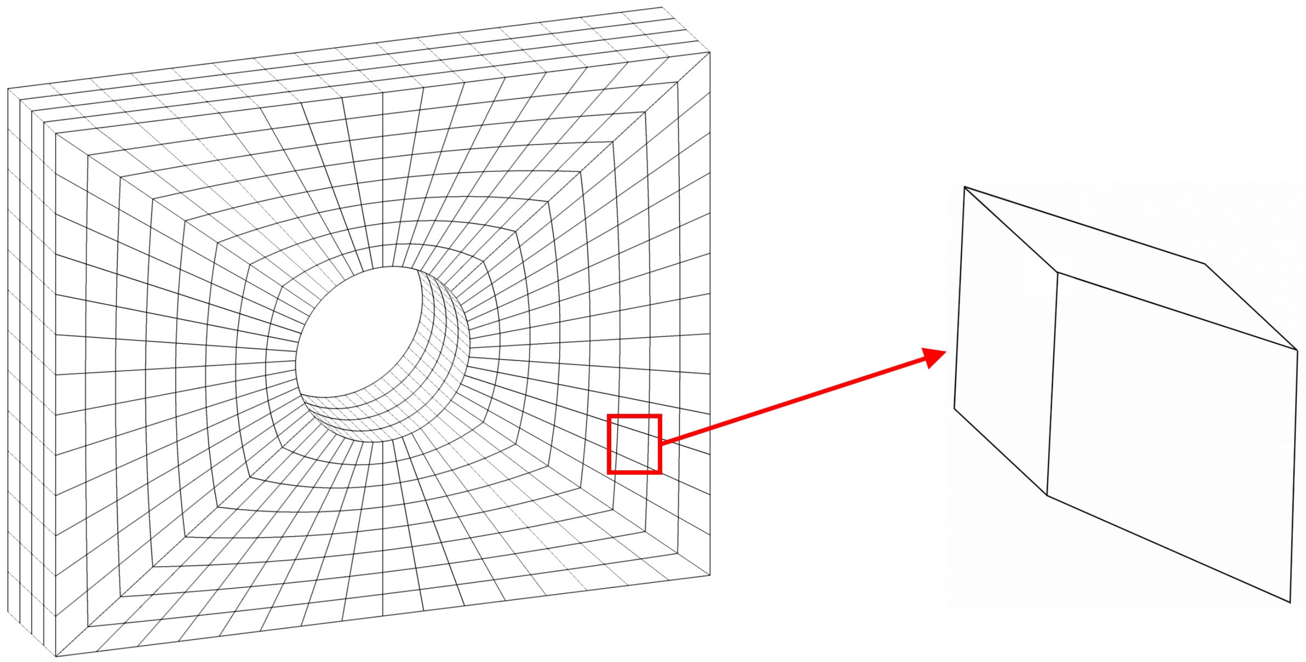
The process of dividing a structure into **smaller parts, called elements**, is called the discretization of a structure in the finite element method.



Model



Mesh with tetrahedral elements



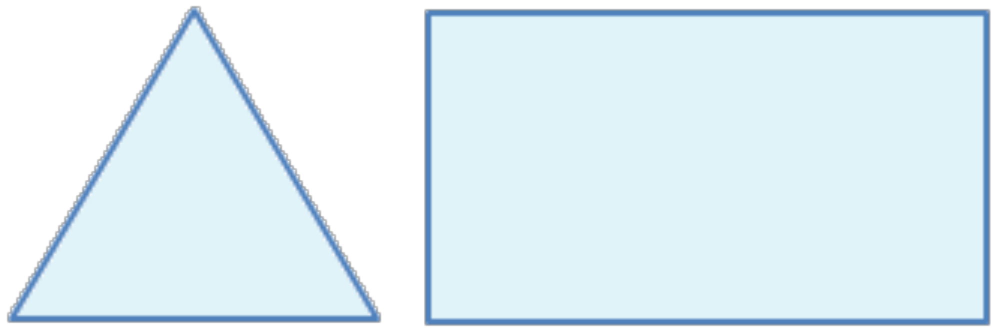
Mesh with hexahedral elements

Common Element Types

The process of dividing a structure into **smaller parts, called elements**, is called the discretization of a structure in the finite element method.

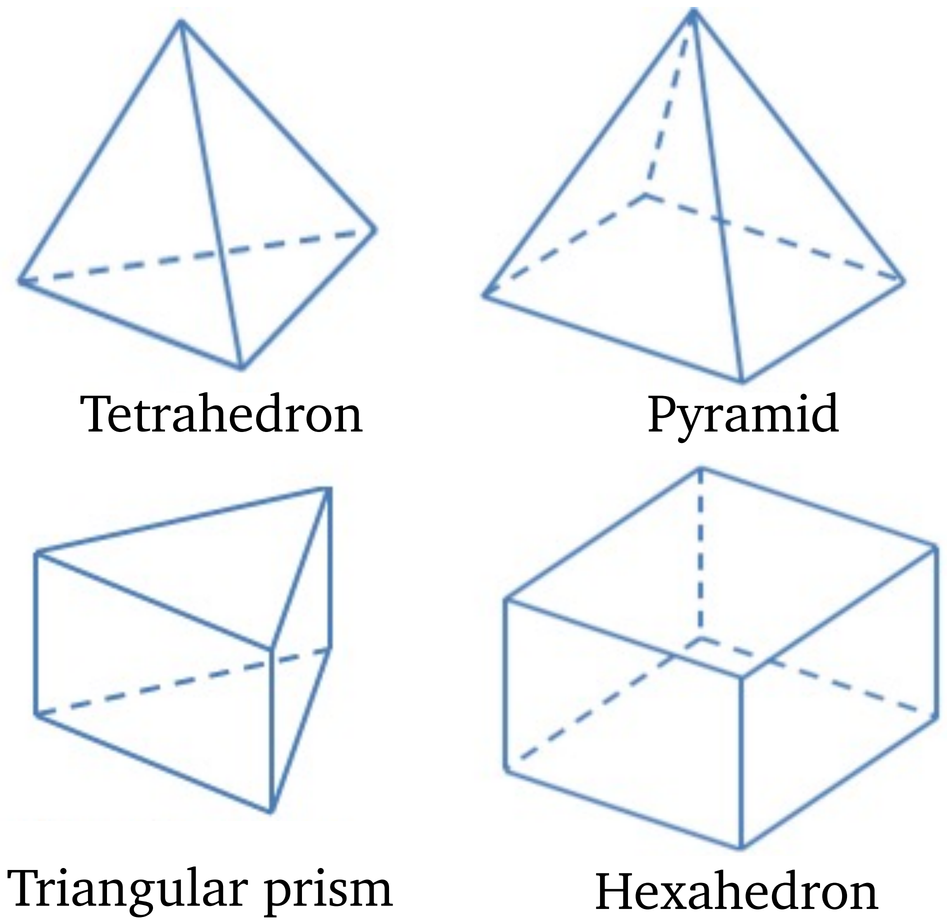
Line

1D element



Triangle Quadrilateral

2D element



3D element

2. Select Interpolation Functions

Interpolation functions are used to provide **an approximation of the unknown solution within an element**. Often, **polynomials are selected as interpolation functions**. The degree of the polynomial depends on the number of nodes assigned to the element.

$$\{u\} = [N]\{d\}$$

$$u = \{u \quad v \quad w\}$$

$$\{d\} = \{u_1 \quad v_1 \quad w_1 \quad u_2 \quad v_2 \quad w_2 \quad \ldots\}$$

$\{u\}$: Displacements at any point inside a finite element.

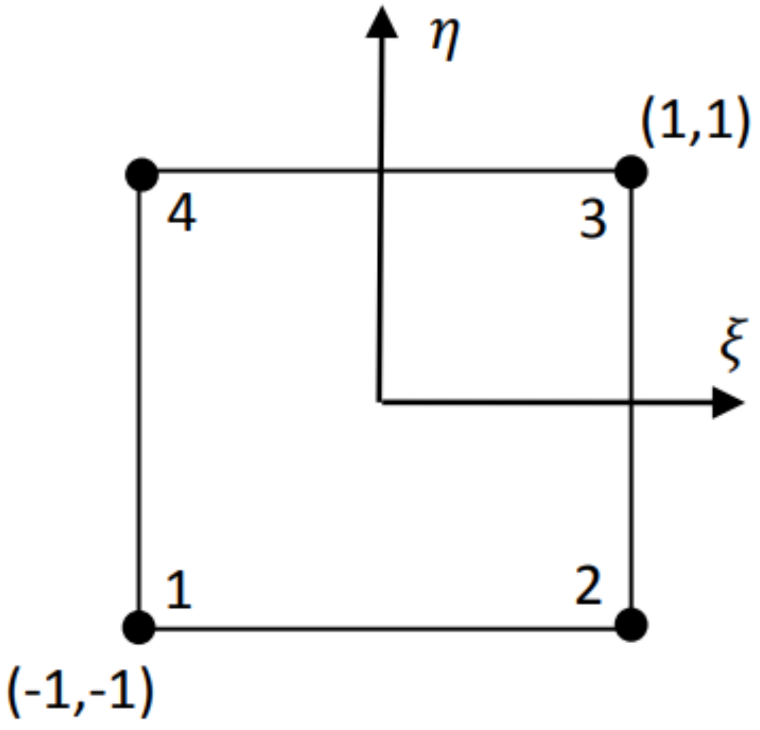
$\{d\}$: Nodal displacements of a finite element.

$[N]$: Shape functions of a finite element.

$$u = \sum N_i u_i$$
$$v = \sum N_i v_i$$
$$w = \sum N_i w_i$$

$$[N] = \begin{bmatrix} N_1 & 0 & 0 & N_2 & \cdots \\ 0 & N_1 & 0 & 0 & \cdots \\ 0 & 0 & N_1 & 0 & \cdots \end{bmatrix}$$

An Example for Shape Function



2D quadrilateral element

General form: $N = A + B\xi + C\eta + D\xi\eta$

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta)$$

$$N_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

$$N_3 = \frac{1}{4}(1+\xi)(1+\eta)$$

$$N_4 = \frac{1}{4}(1-\xi)(1+\eta)$$

3. Find the Element Properties

For this step, different approaches can be used; the most convenient are the variational approach and the Galerkin method.

Element equilibrium equation has the following form: $[k]\{d\}=\{f\}$

$[k]$: Element stiffness matrix.

$\{f\}$: Element force vector.

For example, for 3D element: $[k]=\int_V [B]^T [E][B]dV$

$[E]$: Element elasticity matrix.

$$[E]=\begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}$$

4. Assemble the Element Equations

The individual element nodal equilibrium equations are assembled into global nodal equilibrium equations in this step.

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

$[K]$: System stiffness matrix.

$\{F\}$: System global nodal force vector.

$\{U\}$: System global nodal displacement vector.

5. Solve the Global Equation System

The finite element equation systems are symmetric, positive definite and sparse. Symmetry allows to store only half of the matrix including diagonal entries. Positive definite matrices are characterized by large positive entries on the main diagonal. Solution can be carried out without pivoting. A sparse matrix contains more zero entries than nonzero entries. Sparsity can be used to economize storage and computations.

Solution methods for linear equation systems can be divided into **two large groups**: direct methods and iterative methods. Direct solution methods are usually used for problems of moderate size. For large problems iterative methods require less computing time and hence they are preferable.

6. Compute Additional Results

In many cases, it is necessary to calculate additional parameters. For instance, in mechanical problems, strains and stresses are of interest in addition to displacements, which are obtained after solving the global equation system.

Definition of FEA

The Finite Element Analysis (FEA) is a numerical technique that simulates physical problems using the Finite Element Method (FEM). FEA software is used by engineers to **reduce the number of physical prototypes and experiments and optimize components in their design phase** to develop better products faster while saving on expenses.

General Steps of FEA are follows:

1. Create the model of the physical problem.
2. Define the material properties.
3. Define the element types for the components in the model.
4. Construct the mesh structure of the model.
5. Define the contact settings between the parts in contact with each other.
6. Assign external conditions (load, displacement, temperature, acceleration, …) and boundary conditions.
7. Set the additional required settings for the physical problem.
8. Run the model and check the required results.