Chapter 1: Introduction & Objectives of the FEM

1.1 Equilibrium Equations of an Elastic Solid

*When a body is **subjected** to loads or displacements, **internal forces arise**, which are locally represented by **stresses**. These stresses result in **deformations** that affect the body's **dimensions** and **shape**.*

*Continuum mechanics and structural mechanics deal with the relationships between stresses and deformations, displacements and forces, and stresses and forces, under a set of properly defined boundary conditions. This leads to **differential or partial differential equations**.*

*We provide, in **Appendix A**, a review of these equations, and in the following chapters, we will give briefly a reminder of the basic concepts and classical theories of each type of structure studied.*

1.2 approximate method for resolving the differential equations or the partial differential equations:

*The **resolution** of the differential equations or the partial differential equations by **exact methods** is **easy** only for **a small number** of practical cases (those presenting simple geometries, simple loadings and simple boundary conditions).*

*In most practical cases **this resolution** becomes **difficult if not impossible**, and the use of **approximate methods** is inevitable.*

*We give in **Appendix B** a summary of the most used approximate methods and those most related to the **FEM***

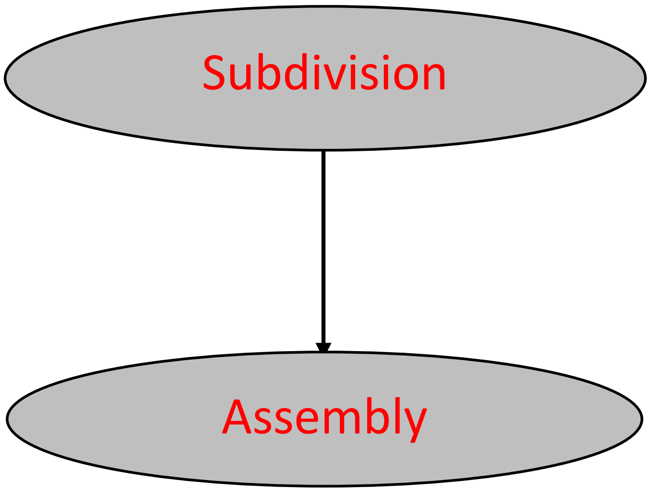
1.3 Definition, objectives and principals of the finite element method:

1.3.1. Definition**: The Finite Element Method (FEM) falls into the category of the approximate methods, and it is a combination of the work of **mathematicians** and the intuition of **engineers

***Mathematicians** have proposed **global methods** applied to the **entire domain** (the entire problem or structure), such as the various **weighted residual methods** and the different **variational methods**.*

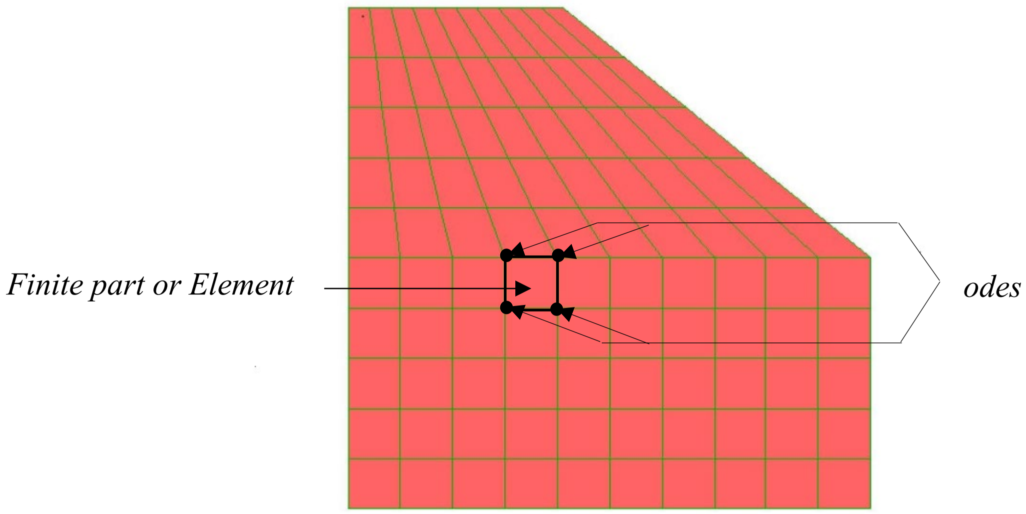
Engineers**, on the other hand, drew an **analogy** between **finite** and **continuous parts** and applied the previous methods to the various **subdomains** or parts (or elements) resulting from a **subdivision** of the entire domain. They did this by replacing the **continuous part** (which is connected to others parts at an infinite number of points) with **a finite part** (connected to others parts at a finite number of points called **nodes**). And from this, the method took its name: **"Finite Element Method (FEM)"

The Finite Element Method can be defined as a global method based on a ***dual process****: subdivision and assembly*



2

1.3.1.1. Subdivision *of the domain into parts (assumed to be finite), and formulation of each element separately: i.e. Replace,* ***the differential equation*** *or the* ***partial differential equation****,* ***D(u) = 0*** *with boundary conditions C(**u**, **f**) = 0, by* ***a system of linear equations*** *{**f**} = [**k**]{**d**}.*



1.3.1.2. Assembly *or Reconstruction of the entire domain by assembling the elements while satisfying certain problem conditions (e.g., in structural mechanics: continuity of displacements and equilibrium of forces at the nodes) to obtain the global system of equations:*

$$\{F\}=[K]\cdot\{D\}$$

For discrete problems, and continuous problems with simple geometry, simple loading, and simple boundary conditions, this method is just one possible variant of resolution. It differs from other methods only by the ease with which the systems of equations that provide the solution to the problem are constructed. However, it is of incomparable utility for problems with complex geometries, loads, boundary conditions, and properties.

1.3.2. General procedure and basic principles of the method:

***1.3.2.1. Subdivide** the structure or the work **into parts** in such a way as to **avoid gaps** or **overlaps**, and to **reproduce** the boundaries **as accurately as possibles**.*

***1.3.2.2 For each part** (or element) assumed to be finite (i.e. connected to the other parts at a finite number of points or nodes), which will be defined by a certain number of nodes, an **approximate function** is **chosen** to represent **the main variable** of the problem (e.g., the displacement field) as a function of its values at the nodes:*

$$\boldsymbol{U}(\boldsymbol{X}) = [\boldsymbol{N}(\boldsymbol{X})] \times \{\boldsymbol{d}\}$$

*Where **X** are the coordinates of an arbitrary point in the element, **U** is the variable or field to be approximated, $\{\boldsymbol{d}\}$ is the vector of the values of **U** at the nodes of the element, called the vector of the **degrees of freedom** of the element, and $[\boldsymbol{N}]$ is the matrix of functions called interpolation functions or shape functions*

a. Approximation by polynomials:

*For an element with "**m**" degrees of freedom related to a component $\boldsymbol{u^k}$ of a displacement field **U**, a polynomial is often chosen to approximate $\boldsymbol{u^k}$, where the number of monomials is equal to the number of degrees of freedom, in the form:* $\boldsymbol{u^k}(\boldsymbol{X}) = \lfloor \boldsymbol{P^k} \rfloor \{\boldsymbol{a^k}\}$

Where $\lfloor \boldsymbol{P^k} \rfloor$ is a polynomial matrix and $\{\boldsymbol{a^k}\}$ is a vector of generalized coordinates. To make writing easier, let's voluntarily omit the superscript index.

$$\boldsymbol{u}(\boldsymbol{X}) = \lfloor \boldsymbol{P} \rfloor \{\boldsymbol{a}\}$$

We then write the boundary conditions:

$$\{\boldsymbol{u}\} = \{\boldsymbol{d}\} = [\overline{\boldsymbol{P}}]\{\boldsymbol{a}\},$$

where $[\overline{\boldsymbol{P}}]$ is the evaluation of $\lfloor \boldsymbol{P} \rfloor$ at the nodes of the element, and $\{\boldsymbol{d}\}$ is the vector of the degrees of freedom of the element, from which we have:

$$\{a\} = [\bar{P}]^{-1}\{d\}$$

So, $$\boldsymbol{u} = [\boldsymbol{P}]\{\boldsymbol{a}\} = [\boldsymbol{P}][\overline{\boldsymbol{P}}]^{-1}\{\boldsymbol{d}\}$$

$$u = [N]\{d\}, \quad with \; [N] = [P][\bar{P}]^{-1}$$

$[N]$ *is called matrix of the* ***interpolation functions*** *or matrix of the* ***shape functions****.*

b. Direct interpolation approximation:

b.1. Lagrange interpolation

$$u = [N]\{u\} \; N_1 u_1 + N_2 u_2 + \ldots + N_m u_m$$

In one dimension the N_i *are interpolation functions determined by the formula:*

$$N_i = \frac{\prod_{\substack{j=1 \\ j \neq i}}^{m} (x - x_j)}{\prod_{\substack{j=1 \\ j \neq i}}^{m} (x_i - x_j)}$$

and having the following property:

$$N_i = 1 \text{ à } u_i \;\; and \;\; N_j = 0 \text{ à } u_j \; for \;\; j \neq i, \; \textit{with}$$

$\prod_{\substack{j=1 \\ i \neq j}}^{m} (x - x_j)$ *: is the product of all monomials* $(x - x_j)$ *from* $j = 1$ *to* $j = m$ *;* $i \neq j$ *and* $\boldsymbol{m}$ *is the number of the degrees of freedom*

b.2. Hermit's interpolation:

In some cases, the derivatives of the unknown variable are used as degrees of freedom (case of bending of beams and plates for example). In these cases, the variable u (the fundamental component of the displacement field) can be represented by:

$$u = N_1 u_1 + N_2 \acute{u}_1 + \ldots + N_p u_1^{(p)} + N_{p+1} u_2 + N_{p+2} \acute{u}_2 + \ldots + N_{2p} u_2^{(p)} + \ldots + N_{(q-1)p+1} u_q + N_{(q-1)p+2} \acute{u}_q + \ldots + N_{qp} u_q^{(p)}$$

Where the exponent in parentheses denotes the order of derivation, which gives:

$$u^{(p)} = N_1^{(p)} u_1 + N_2^{(p)} \acute{u}_1 + \ldots + N_p^{(p)} u_1^{(p)} + N_{p+1}^{(p)} u_2 + N_{p+2}^{(p)} \acute{u}_2 + \ldots + N_{2p}^{(p)} u_2^{(p)} + \ldots + N_{(q-1)p+1}^{(p)} u_q + N_{(q-1)p+2}^{(p)} \acute{u}_q + \ldots + N_{2p}^{(p)} u_q^{(p)}$$

We then have $(q \times p)$ *conditions for constructing each of the deformation function* $\boldsymbol{N_i}$*,*

$$N_i = 1 \; at \; U_i \;\; and \;\; N_j = 0 \; at \; U_j \;\; for \; j \neq i$$

$$N_i^{(p)} = 1 \; at \; U_i^{(p)} \;\; and \;\; N_i^{(p)} = 0 \; at \; U_i^{(p)} \; for \; j \neq i$$

Which brings us, in the one-dimensional case, to writing each N_i *in the form:*

$$N_i = a_0 + a_1 x + a_2 x^2 + \ldots + a_{pq-1} x^{pq-1}$$

The generalization of these two one-dimensional interpolations to two and three dimensions is achieved by combining the one-dimensional interpolations in the considered directions and will be addressed in the study of planar and three-dimensional elements.

***1.3.2.3: Use this approximated field to formulate the element**:*

Replace the differential or partial differential equations of the element by a system of linear equations:

$$\{\boldsymbol{f}\}^{\boldsymbol{i}} = [\boldsymbol{k}]^{\boldsymbol{i}}\{\boldsymbol{d}\}^{\boldsymbol{i}},$$

*Where $[k]^i$ is a matrix function of the geometric and physical properties of the element « **i** » called stiffness matrix, and $\{d\}^i$ is a vector of the degrees of freedom of the element written in an element specific coordinate system called the local coordinate system.*

The formulation methods that we will use, and which will be presented during the study, by finite elements, of the different types of structures approached, are:

- *Direct rigidity method: combination of groups of equations of the problem, for example in structural mechanics it is the combination of three groups of equations: Geometric equations (or displacement-deformation equations), Behavior equations (or stress-deformation equations) and equilibrium equations*
- *Principle of virtual work*
- *Weighted residue methods applied to differential equations or partial derivatives of the problem*
- *Variational methods applied to potential energies and complementary energies*

In most of these methods we need the strain and stress fields, so we must express them as a function of the degrees of freedom of the element:

the strain field $\boldsymbol{\varepsilon}$ can be determined from the geometric relations (displacement - strain relations) and which is presented as a differentiation of the displacement field:

$$\boldsymbol{\varepsilon} = diff(u) = diff\ ([N]\{d\}) = diff[N]\{d\}$$

$$\boldsymbol{\varepsilon} = [\boldsymbol{B}]\{\boldsymbol{d}\} \quad with\ [\boldsymbol{B}] = \boldsymbol{diff}[\boldsymbol{N}]$$

the stress field $\boldsymbol{\sigma}$ can be determined from the behavior relations:

$$\boldsymbol{\sigma} = [\boldsymbol{C}]\boldsymbol{\varepsilon} = [\boldsymbol{C}][\boldsymbol{B}]\{\boldsymbol{d}\}$$

1.3.2.4 Assembly** of the elements by satisfaction of certain conditions (here balance of forces and continuity of displacements) to reconstruct the initial domain or entire structure, which leads to obtaining a global system of equations for the complete structure written in a coordinate system chosen for the entire structure. This is called a global coordinate system**:

$$\{\boldsymbol{F}\} = [\boldsymbol{K}]\{\boldsymbol{D}\}$$

Where:

$[K]$ *is the global stiffness matrix of the entire structure.*

$\{D\}$ *is the global degrees of freedom vector of the entire structure (here, the displacements at the structure's nodes, but in general, it is the vector of the primary variable values of the problem at the nodes).*

$\{F\}$ *is the vector of known values of the problem (here, the forces at the structure's nodes).*

***1.3.2.5. Resolution** of the global system and determination of the unknown quantities at the global level and at the level of each element*

The Solving the system $\{\boldsymbol{F}\} = [\boldsymbol{K}]\{\boldsymbol{D}\}$*after introducing the boundary conditions gives us the vector.*$\{\boldsymbol{D}\}$**.**
For each element "i", we deduce from$\{\boldsymbol{D}\}$ *the vector* $\{\boldsymbol{D^i}\}$ *in the global coordinate system and* $\{\boldsymbol{d^i}\}$ *in the local coordinate system, then from the system* $\{\boldsymbol{f}\}^{\boldsymbol{i}} = [\boldsymbol{k}]^{\boldsymbol{i}}\{\boldsymbol{d}\}^{\boldsymbol{i}}$ *, we determine the vector* $\{\boldsymbol{f}\}^{\boldsymbol{i}}$ *at the nodes as well as the other gradients of these vectors (In structural mechanics* $\{\boldsymbol{f}\}^{\boldsymbol{i}}$ *represents the internal nodal forces, the gradients of* $\{\boldsymbol{f}\}^{\boldsymbol{i}}$*and* $\{\boldsymbol{d}\}^{\boldsymbol{i}}$*i are the stresses and deformations).*

1.3. 3. Errors in the finite element method

1.3.3.1 Sources of errors:

Let's consider a plate subdivided into 2 elements.

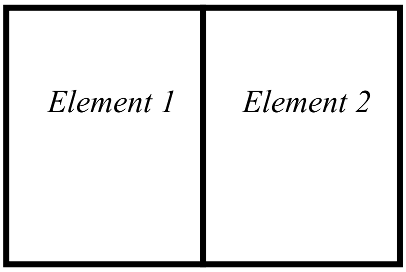
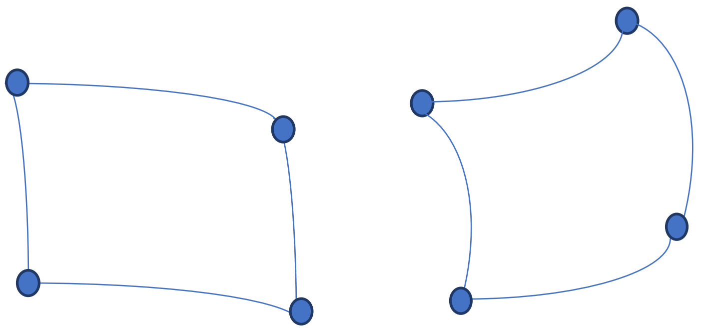


Plate before deformation

As during the assembly, the conditions of the problem (here the continuity of the displacements and the equilibrium of forces) are only guaranteed at the nodes of the elements, on the common boundary, one or both of these conditions therefore may not be satisfied.

1st Source of errors: Failure to satisfy displacement continuity condition:



Element 1 after deformation *Element 2 after deformation*

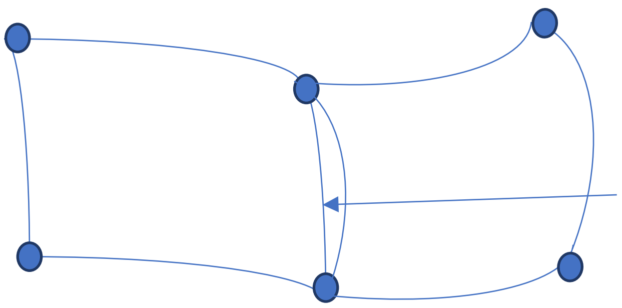
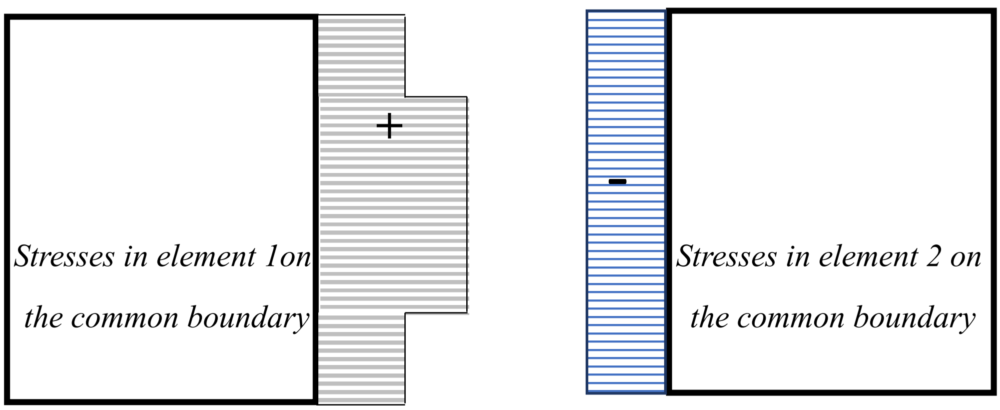
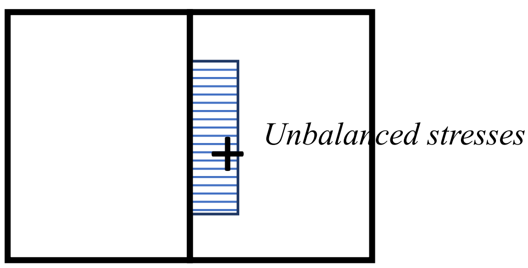


Plate after deformation

2nd source of errors Failure to satisfy force equilibrium:

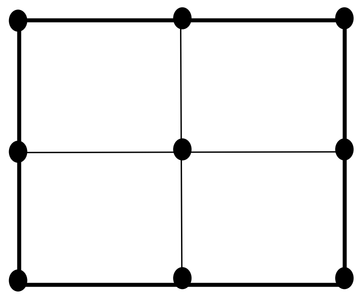




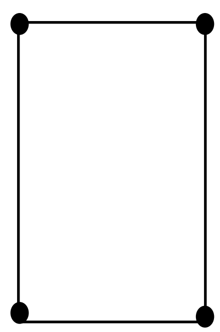
1.3.3.2 Improvement of the FEM solution by reducing the error

Two solutions are possible:

'h' Solution: *Mesh Refinement (Reducing Element Size)*

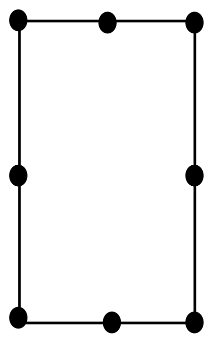


'P' Solution: *satisfaction of the conditions of the problem at a certain number of points of the common boundary by using higher order elements*



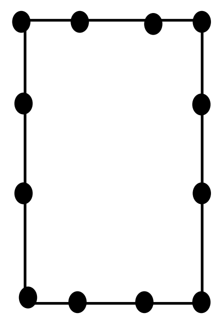
Rectangular element

Simple with 4 Nodes



Rectangular element

Quadratic with 8 Nodes



Rectangular element

Cubic with 12 Nodes

1.3.4. FEM General Programs

A general program is a program that allows the resolution of a wide range of problems, it is composed of the following groups of modules:

Data input.	Modules or Sub-programs allowing to specify the problem data (for example: geometry, mesh, geometric and physical characteristics of the elements, loading, support conditions etc.)
Elements Library	Modules or sub-programs allowing the formulation of each element covered by the program (constructing its stiffness matrix [***k***], its force vector {f}, etc.).
Assembly and resolution	Modules or sub-programs allowing the construction of a global system $\{\boldsymbol{F}\} = [\boldsymbol{K}]\{\boldsymbol{D}\}$ by assembling the elements, the introduction of boundary conditions, and the resolution of the reduced system obtained to determine the values of the main variable of the problem at the nodes of the subdivision
Restitution calculations	Modules or subprograms for determining the values of other variables in the problem (for example: deformations, stresses and internal forces, etc.)
Presentation of the results	Modules or sub-programs allowing the presentation of results graphically or in the form of tables to facilitate their use