MASTER Mécanique et Ingénieries

CONTINUUM MECHANIC AND FINITE ELEMENT METHOD

Finite elements

4TME701U | Yann.LEDOUX@u-bordeaux.fr

Content

1 Introduction

1.1 Background

The simulation of the physical phenomena is a key issue of engineers. The main motivation is the understanding of the system behaviors.
Most of the time, the setup of simulations have to answer these specific questions:

- What must be considered to carry out a study/ which physical phenomena / which modeling type....
- Where to isolate the system
- Which and How to introduce boundary conditions

The finite element modeling activity starts from real physical observations, to make approximations on physical fields (temperature, stress fields...) and simplifications of real geometry as illustrated in figure 1.

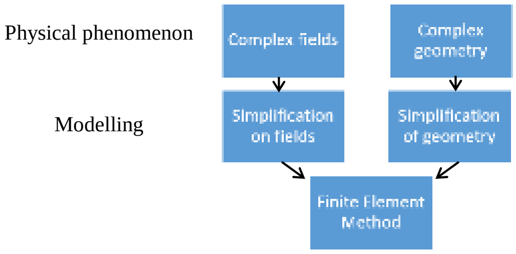


Figure 1 : main steps to follow during the finite element modeling approach.

Basically, the definition of problem starts from the observation of particular physical phenomena, next the engineers define the mathematical modeling most suited to the observations (usually based on differential equations). Then, classical approach to solve such formulation is based on simplification of the formulation (discrete modeling) and finally, the numerical code is defined (through matrix system) to make possible the resolution of the problem by the means of computing methods.

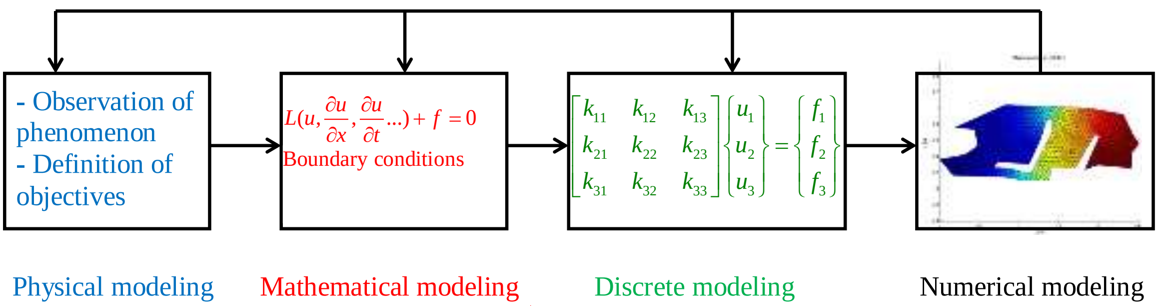


Figure 2 : illustration of the simplification steps ; from the mathematical formulation to the numerical resolution.

The different simplifications use to make more tractable the problem lead to several approximations and error propagation. One of the main motivation is to estimate their values.

The main classical modeling formulations used for the physical study of systems are listed in the following table. The developed formulation in this **course assumed a stationary state of the problem and discrete formulation for which an equilibrium equation could be written** (1st Newton's law).

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

with {F} : load vector, [K] : stiffness matrix and {u} displacement vectors.

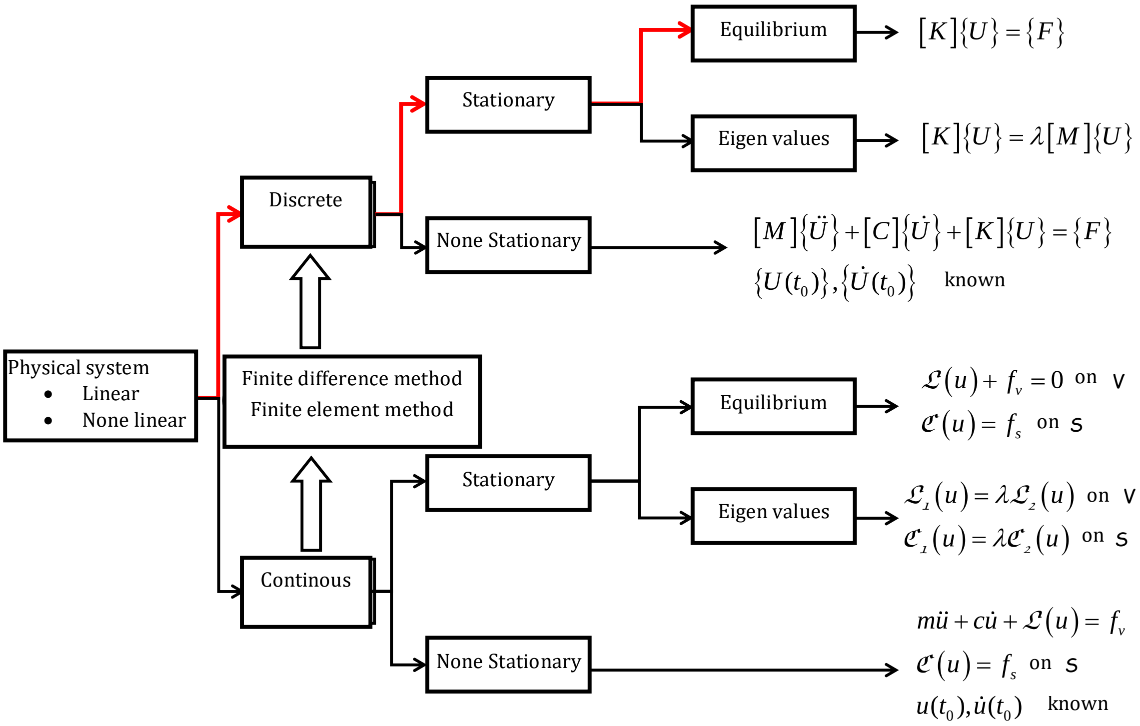


Figure 3 : classical modeling formulation for physical studies.

1.2 Statements

The finite element method is classically used for a wild range of applications of the engineering studies. It could be cited :

- The definition of design architecture of products and systems
- The dimensioning of parts.
- The optimization of process
- The understanding of particular and complex phenomena (machining operation, flow of material…)

According to the studied phenomena, the numerical simulation could be :

- A static simulation in case of:
  - Structural computation and dimensioning
  - Strain and stress computation according to the loading
  - Vibration study
    - Modal analysis (principal modes identification)
    - Noise study and propagation
- A dynamic simulation for:
  - Crash test modeling
  - Process simulation
    - Forming process, machining operations…
- Thermo-mechanical simulation:
  - Modification of mechanical property of materials according to thermal variation
  - Process simulation
    - Forming process, machining operations, additive manufacturing process…

However, each of these approaches and methodologies require:

- to take precautions regarding the computed results (i.e. it is not enough to see only the color gradients),
- to be able to estimate the expected results (order of magnitude), identify the expected mechanical quantities
- to set up validations by comparing the results with experimental tests
- and finally, to use simplified models from simple analytical cases to estimate the expected results.

The main difficulties come from:

- the modeling and characterization of the material
- the geometrical modeling of the problem (mesh)
- the choice of geometrical characterization (type of elements and dof) and model assumptions (symmetry, displacement)
- the contact management / the application of effort / the pressure loads …

2 Deductive definition of unidimensional finite element: the truss element

2.1 General consideration

A truss is one of the simplest and widely used structural elements. It corresponds to a straight bar designed to take only axial forces and it deforms only in its axial direction. Some typical conditions are required concerning the cross-section (should be much smaller than the axial dimension). Such elements are commonly used for the modeling of skeletal type of truss structural systems in two or three-dimensional space.

Figure 4 : architectural illustration of truss structure.

2.2 Stiffness definition

The main motivation is to deduce from the basic concept the characteristic of truss elements. To do this, three hypotheses is required :

- H1 : **linear behavior of material**: the behavior of the material assumes a linear relation between the stress and the deformations (also called internal linearity condition).
- H2 : **small deformations :** the deformations are modeled by linear relation between stress and strain computation (also called external linearity condition).
- H3 : The study is supposed to be a **static problem.**

Following these hypotheses, it could be observed a linear relation between the stress and the displacements. That means that for a structure subject to boundary conditions (f_i, T_i^d, u_i^d) [1], if the solution is (σ_{ij}, u_i); then, for the limit condition (λf_i, λT_i^d, λu_i^d), it could be deduced that the solution become ($\lambda\sigma_{ij}$, λu_i).

[1] with f_i : internal load, T_i^d: external load, u_i^d : displacement.

What is the consequence of these hypotheses (linearity and proportionality) on the behavior an one-dimensional structure? Let consider a structure subject to a traction load belonging to the axis of the structure as shown in figure 5.

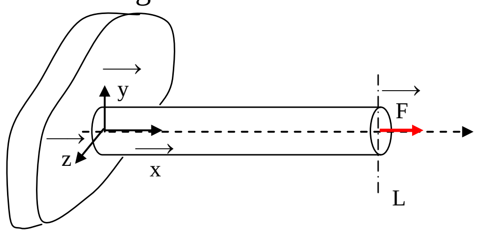
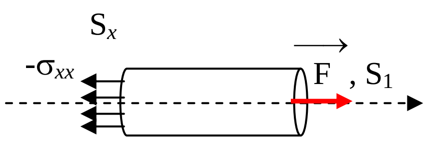


Figure 5 : illustration of the one-dimensional structure subject to traction load

Assuming a virtual portion of the structure belonging the x-axis :



Remark : it is assumed that for all points M of the section S_x, the stress is expressed by

$$\vec{T}(M,-\vec{x}) = -\sigma_{xx}\,\vec{x}$$

The equilibrium could be written as follow:

$$\mathbf{F - \sigma_{11}.S = 0}$$

Thus, $\sigma_{11} = \dfrac{F}{S}$

With S is the cross section of the structure.

The constitutive equations could be reduced to Hooke's law for 1D solids (case of one-dimensional problem, excluded the transverse deformation):

$$\sigma_{11} = E.\varepsilon_{11} \tag{1}$$

With E corresponds to the Young's modulus.

Remark: it could be observed the linear relation assumed in the previously.

The strains are then defined by :

$$\varepsilon_{11} = \frac{du_1(x)}{dx} \tag{2}$$

where : $\dfrac{du_1(x)}{dx} = \dfrac{\sigma_{11}}{E}$ **= constant (since σ_{11} and E are constant values).**

Thus, $u_1(x) = \dfrac{\sigma_{11}}{E}x + c^{st}$

If the boundary conditions are introduced into the previous equation, it could become:

$$u_1(0)=0 \text{ d'où } u_1(x)= \frac{\sigma_{11}}{E}x$$

And for x = L

$$u_1(L) = \frac{FL}{ES}$$

Finally, the particular solution of the problem is:

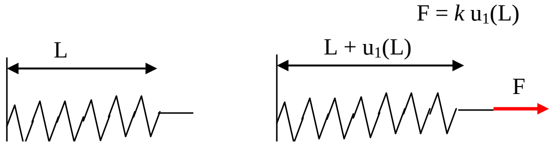
$$u_1(x) = x.\frac{F}{ES} \qquad (3)$$

$$\sigma_{11} = \frac{F}{S}$$

The proportionality between the load, F, and the solution (σ_{11}, $u_1(x)$) is validated.

With equation (3) and for x=L shows that $F = E\,\frac{S}{L}\,.\,u_1(L)$

It could be observed that the behavior of the truss corresponds to an axial spring with a stiffness defined by the particular ratio: $E.\frac{S}{L}$



If we want to change the stiffness value of the structure, two kinds of modifications could be made:

- On the material, modification of E
- On the geometry, $\frac{S}{L}$

Remark

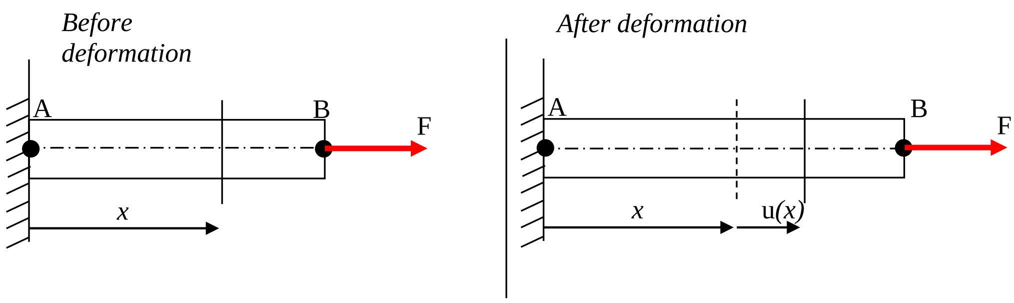
It could be noticed that the linear relations $\sigma_{11}=E.\varepsilon_{11}$ and $\varepsilon_{11} = \frac{du_1(x)}{dx}$ leads to proportionality between external load F and the displacement field u(x). More to this, the stiffness is not a function of the load and is directly defined by the material and the geometry of the structure.

3 Hypothesis of the displacement field and the stiffness matrix of the truss element

The construction of a finite element is always based on an assumption about the displacement field. In some cases, this assumption corresponds to those made in the theory of the continuum mechanic such as the theory of beams, plates or shells. This section is dedicated to the construction of the truss element.

3.1 Beam subjected to traction

The Euler-Bernouilli assumption for thin beams assumes that's that the cross section remains perpendicular to the median line of the beam.



Based on this assumption, we can write that:

$$\vec{u}\,(x,y,z)=\mathrm{u}(x).\,\vec{x}$$

The relation between F and u(x) is then:

$$\varepsilon_{xx} = \frac{\mathbf{du}}{\mathbf{d}x}\ ;\ \ \varepsilon_{xy} = \varepsilon_{yy} = \ldots = 0$$

$$\sigma_{xx} = \mathbf{E}\,\frac{\mathbf{du}}{\mathbf{d}x}$$

After the integration on a straight section

$$\int_s \sigma_{xx} = \int_s \mathbf{E}\,\frac{\mathbf{du}}{\mathbf{d}x}\,\mathbf{ds}$$

$$\mathbf{F} = \mathbf{ES}\,\frac{\mathbf{du}}{\mathbf{d}x}$$

$$\textbf{Soit}\quad \frac{\mathbf{du}}{\mathbf{d}x} = \frac{\mathbf{F}}{\mathbf{ES}}$$

For a constant value of F and for a constant cross section, this leads to:

$$\int_0^x \mathbf{du} = \int_0^x \frac{\mathbf{F}}{\mathbf{ES}}\,\mathbf{dt}$$

$$\mathbf{u}(x) - \mathbf{u}(0) = \frac{\mathbf{F}}{\mathbf{ES}}\,.\,x$$

$$\mathbf{u}(x) = \frac{\mathbf{F}}{\mathbf{ES}}\,.\,x + \mathbf{u}(0)$$

Since the displacement field has a linear behavior, it could be defined by two constants. These constants could be for example the extremal displacement of the points 1 and 2 (called the nodes). So:

u(0)=u1 et u(L) = u2

The relation of the displacement as a function of the position belonging the element could be defined by the following equation:

$$\mathbf{u}(x) = \mathbf{u}_1\left(1 - \frac{x}{L}\right) + \mathbf{u}_2\,.\,\frac{x}{L}$$

Which could be written as :

u(x) = N1(x).u1 + N2(x).u2

where Ni(x) is called the <u>shape function</u>.

3.1.1 Conclusion

The Bernouilli's hypothesis or more generally, regarding the displacement field of structure, it is possible to fully determine the mathematical relation describing it. The switch from continuous (displacement field) to discrete (two nodal displacements u1 and u2) is also possible since the displacement is defined through a finite number of nodal displacements.

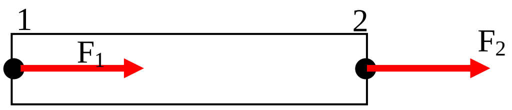
It could also be noticed that from the displacement field, the deformations and stresses in the element are defined by:

$$\varepsilon_{xx} = \frac{u_2 - u_1}{L}$$

$$\sigma_{xx} = E\,\frac{u_2 - u_1}{L}$$

3.2 Stiffness matrix

From the displacements u_1 and u_2, the stress state can be computed. By the integration of the stress function, the value of load F_1 and F_2 are computed. A relation can be expressed between the loads (F_1, F_2) and nodal displacements (u_1, u_2).
This linear relation (expressed previously) can be written through a matrix formulation. This particular formulation is used to express the “stiffness matrix” of the truss.



Equilibrium condition:

$$F_1 + F_2 = 0 \qquad (4)$$

Relation between load and displacement (F : normal load = F_2) :

$$F_2 = \frac{ES}{L}(u_2 - u_1) \qquad (5)$$

Reporting the result of (4) in the equation (5), we obtained:

$$F1 = \frac{ES}{L}(u_1 - u_2)$$

Through matrix relation:

$$\begin{pmatrix} F_1 \\ F_2 \end{pmatrix} = \begin{pmatrix} \frac{ES}{L} & \frac{-ES}{L} \\ \frac{-ES}{L} & \frac{ES}{L} \end{pmatrix} . \begin{pmatrix} u_1 \\ u_2 \end{pmatrix}$$

Thus :

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

With {F} : vector of nodal forces
[K] : stiffness matrix.
{u} : vector of nodal displacements.

Remark :

- The stiffness matrix is a symmetric matrix.
- The matrix K is a singular matrix (determinant is zero).

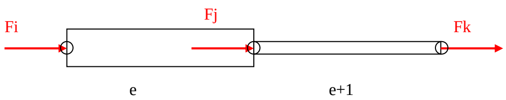
Thus, if the beam is subjected to loads F_1 et F_2:

- If $F_1 \neq - F_2$: the equilibrium equations are not verified and there is no solution.
- If $F_1 = - F_2$: there are an infinite number of solutions ; if a couple of displacements (u_1, u_2) is the solution, then the couple (u_1+a, u_2+a) is also a solution $\forall$ a. That means that the solutions are defined for a rigid solid displacement.
  To find a particular solution of the problem, it is necessary to impose the value of *a* and then, the particular computations of u_1 and u_2 are possible.

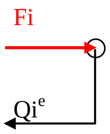
If a displacement is imposed as a boundary condition on nodes (e.g. on u_1), then it is not possible to define an external load on the same node (e.g. on F_1) that mean that the load (F_1) becomes an unknown parameter of the problem to solve.

4 Assembly procedure of elementary stiffness matrix

Let suppose a structure made up of two finite elements subjected to three external loads as shown in figure below.

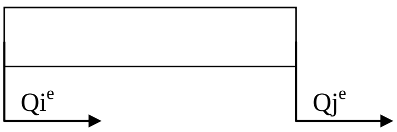


With F_i, F_j, F_k : external loads applied in every node
And Qi^e *the internal loads of the element e*



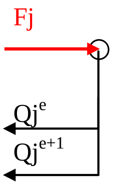
Equilibrium of the node i

$$-Qi^e + Fi = 0$$



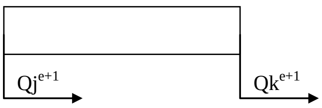
Equilibrium of the element e

$$Qi^e + Qj^e = 0$$



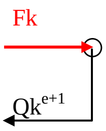
Equilibrium of the node j

$$-Qj^e - Qj^{e+1} + Fj = 0$$



Equilibrium of the element e+1

$$Qj^{e+1} + Qk^{e+1} = 0$$



Equilibrium of the node k

$$-Qk^{e+1} + Fk = 0$$

We can sum up the different relations:

$$Qi^e = Fi \tag{6}$$

$$Qj^e + Qj^{e+1} = Fj \tag{7}$$

$$Qk^{e+1} = Fk \tag{8}$$

If we introduce the previous relation between the stiffness and the nodal displacements, we can write:

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

Assuming that the nodal displacement of the node f is called qf,

$$\begin{bmatrix} Qi^e \\ Qj^e \end{bmatrix} = \begin{bmatrix} Kii^e & Kij^e \\ Kij^e & Kjj^e \end{bmatrix} . \begin{bmatrix} qi^e \\ qj^e \end{bmatrix}$$

$$\begin{bmatrix} Qj^{e+1} \\ Qk^{e+1} \end{bmatrix} = \begin{bmatrix} Kjj^{e+1} & Kjk^{e+1} \\ Kjk^{e+1} & Kkk^{e+1} \end{bmatrix} . \begin{bmatrix} qj^{e+1} \\ qk^{e+1} \end{bmatrix}$$

Then, into the equations (6 to 8),

$$Kii^e.qi + Kij^e.qj = Fi$$

$$Kij^e.qi + [Kjj^e + Kjj^{e+1}]\,.qj + Kjk^{e+1}.qk = Fj$$

$$Kjk^{e+1}.qj + Kkk^{e+1}.qk = Fk$$

Finally, through matrix relation:

$$\begin{bmatrix} Fi \\ Fj \\ Fk \end{bmatrix} = \begin{bmatrix} Kii^e & Kij^e & 0 \\ Kij^e & Kjj^e+Kjj^{e+1} & Kjk^{e+1} \\ 0 & Kjk^{e+1} & Kkk^{e+1} \end{bmatrix} . \begin{bmatrix} qi \\ qj \\ qk \end{bmatrix}$$

5 Introduction of the boundary conditions

In the previous section, we have seen how to assemble the different elementary stiffness to express the global stiffness of a structure (K). The relation is:

$$\{F\} = [K]\,\{q\}$$

With {F} : vectors of nodal loads
{q} : vector of nodal displacements
[K] : stiffness matrix of the structure.

This relation corresponds to the integration of both the geometry shape of the structure and the material property. To find the final equilibrium of the structure, it is necessary to introduce the boundary conditions.

Two kind considerations are possible:

- Either, the nodal load is unknown and then displacement is defined
- Either, the nodal load is applied and then, the displacement is deduced.

5.1.1 Case of applied loads

It is supposed that the load is only applied and concentrated to nodes. The load directly corresponds to a particular value of the vector of nodal loads ({F}).

5.1.2 Case of nodal displacements

As previously defined, the solution of the finite element problem ({F}=[K]{q}) is defined whatever the solid displacement of the structure (displacement field without any deformation). The rigid body displacement corresponds to:

- 3Rotations
- 3Translations.

To define a particular solution of the finite element problem, it is necessary to avoid these rigid displacements by imposing some motions (translation and/or rotation).

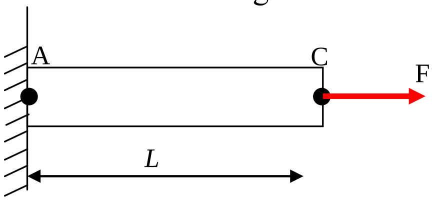
5.1.3 Method for the resolution of finite element problem

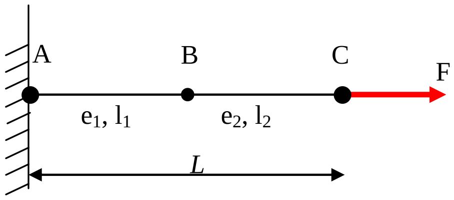
- If, in a particular node, the structure is constrained by supports (e.g. external parts or frame). The displacement is then imposed and the loads correspond to a resultant of the problem. This value will be computed.

- If, in a particular node, a nodal load is applied, the strategy of resolution follow this procedure :
  - 1. The resolution of the matrix relation is made in which only the equation without null degree of freedom and imposed nodal load is considered.
    - In this case, a cancellation of rows and column in stiffness matrix is made, hence K becomes Symmetric Positive Definite Matrix.
  - 2. Then, the contact reaction of the problem is computed from the equations assuming the nodal displacement supposed to be null (the use of suppressed equation at the previous step).

5.1.4 Illustration: case of fixed bar subject of external load

Consider a bar of uniform cross-sectional area shown in figure. The bar is fixed at one end and is subjected to an horizontal load of F at the free end. The beam is made of an isotropic material with Young's modulus E.





1. Set up an analytical resolution of the problem (based on continuum mechanic equation).
2. Set up a finite element resolution assuming a model composed of two truss elements.

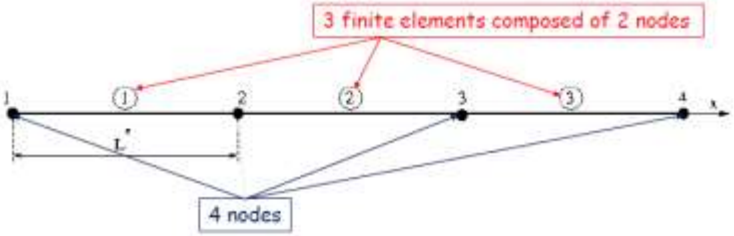
- Write the stiffness matrix for every element.
- Assemble the two stiffness matrix to define the stiffness of the structure
- Express the problem to be solved and solve it.

5.1.5 Modification of the previous problem

Consider the same problem previously studied and add an external load on the node B called Fb. Follow the same steps to solve the problem (analytical formulation and finite element approach).

6 Assembly procedure, a general formulation

General case with several elements

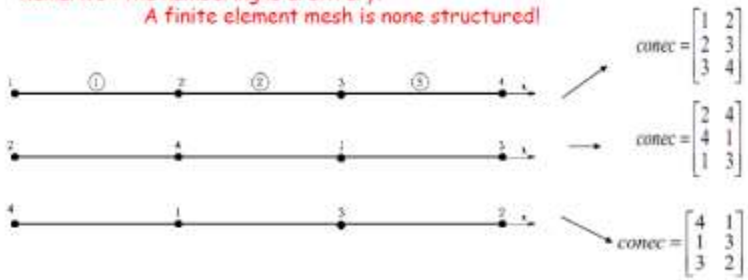


- A Finite element mesh is described by two kind of tables:
  - … the nodal coordinates: $vcor = [x_1 \quad x_2 \quad x_3 \quad x_4]$
  - … the connectivity: $conec = \begin{bmatrix} 1 & 2 \\ 2 & 3 \\ 3 & 4 \end{bmatrix}$

For every element: a list of nodes

Remarks on the mesh

Remark 1 : the numbering is arbitrary.
A finite element mesh is none structured!

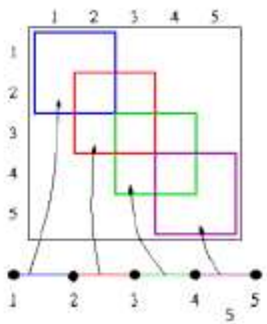


Remark 2 : the length of element could be different

Assembly phase

- The assembly phase follow this steps:
  - To assemble all elementary stiffness matrix into the global stiffness matrix of the structure $[K]$
  - To assemble all elementary vector into a unic vector of loads $\{F\}$

Two assembly technic could be used !



Assembly by extension (not so common)

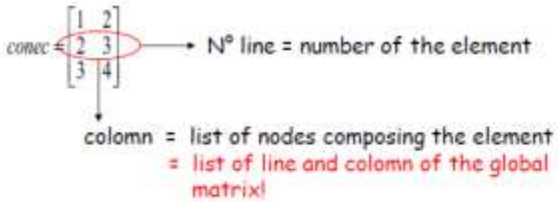
- The principle is to: increase the dimension of every elementary matrix and vectors to the dimensions of the global stiffness matrix
  - Example:

$$\frac{ES_1}{L_1}\begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ 0 \\ 0 \end{Bmatrix} + \frac{ES_2}{L_2}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & -1 & 0 \\ 0 & -1 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ u_3 \\ 0 \end{Bmatrix} + \frac{ES_3}{L_3}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ 0 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}$$

Disadvantage: all elementary matrices have the size of the global matrix
→ memory limitation on computer

Assembly by projection

- Principle: it consists in locating the "zone" of the global matrix where the elementary matrix will be projected
  - ➢ This zone has the same dimensions than the elementary matrix.
  - ➢ Tool to identify the contribution of elementary stiffness matrix: the connectivity table « conec »



- The process remains the same for the elementary vector of loads.

7

Technique d'assemblage par projection

General steps:

- For every elements of the structure:
  - ➢ Definition of $[K^e]$ and $\{F^e\}$
  - ➢ Extraction of the connectivity between elements (numbering of nodes)
  - ➢ Identification in $[K]$ and $\{F\}$ of the modification on line and colomn
    1. Projection of $[K^e]$ into $[K]$
    2. Projection of $\{F^e\}$ into $\{F\}$
- Next element
- Introduction of boundary conditions

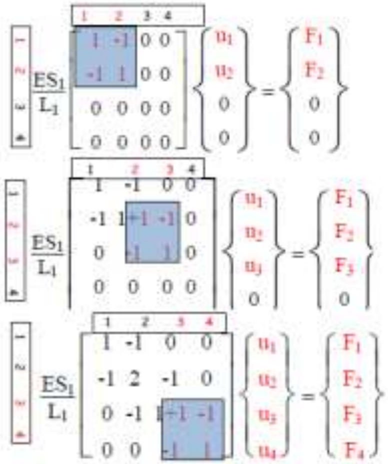
8

Illustration on mesh with 3 elements

- Assembly of the element 1
  ➢Conec(1,[1 2])=[1 2]
  Nber element — Nber of colomns — List of nodes

Assembly of the element 2
Conec(2,[1 2])=[2 3]

Assembly of the element 3
Conec(3,[1 2])=[3 4]



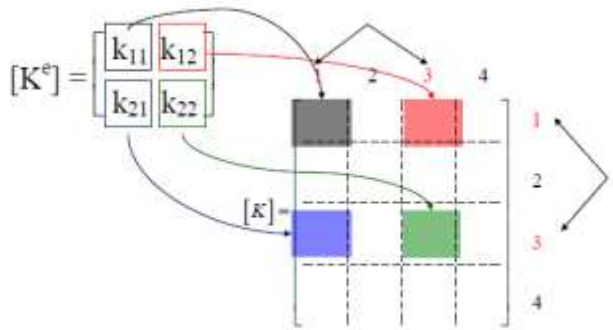
Remark : to simplify, it is assumed that $L_1 = L_2 = L_3 = L$ the same for the surfaces.

9

Particular case of assembly

None consecutive nodes

Example : conec(e, [1 2]) = [1 3]



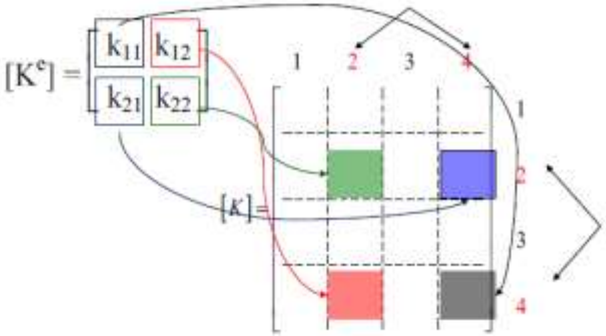
Put stiffness contribution keeping the respective relative positions!

10

Particular case of assembly

None consecutive nodes

Example : conec(e, [1 2]) = [4 2]



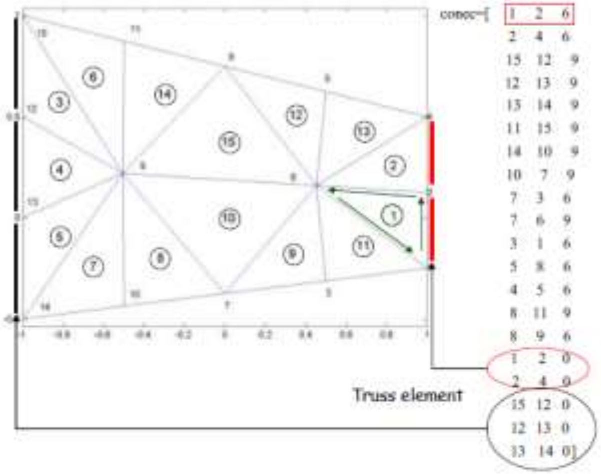
Put stiffness contribution keeping the respective relative positions!

11

Case of 2D mesh

•

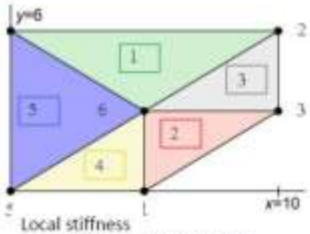
vcor=[1.0000	-0.2500
1.0000	0.1250
0.5000	-0.3125
1.0000	0.5000
0.5000	0.6250
0.4512	0.1593
-0.0000	-0.3750
-0.0000	0.7500
-0.5054	0.2191
-0.5000	-0.4375
-0.5000	0.8750
-1.0000	0.5000
-1.0000	0
-1.0000	-0.5000
-1.0000	1.0000]



Convention on the reading direction of the nodes = trigonometric order!

12

For more complex assembly



Local stiffness matrix

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix}$$

Connec table

$$\begin{bmatrix} 2,4,6 \\ 1,3,6 \\ 3,2,6 \\ 5,1,6 \\ 5,6,4 \end{bmatrix}$$

13

7 General construction of finite element

In the previous sections, a relationship between displacements and nodal loads of a structure has been established. This requires that all boundary conditions (displacement or loads) are located at the nodes. So how to take into account, for example, the case of distributed loads over a structure?

The technique consists in setting up a discretization of the loads in order to reduce it to an equivalent problem for which the loads are located on the nodes of the structure.

To do this, an equivalence criterion between the continuous and discrete problem should be defined. The notion of equivalence between physical quantities of a different nature is based on an energy criterion. Thus, two types of energy are considered:

- The deformation energy
- The work of external loads

$$\Pi(u_i) = \int_V \left(\frac{1}{2} C_{ijkl}\varepsilon_{ij}\varepsilon_{kl} - \sigma_{ij}^0 \varepsilon_{ij} \right) dV - \int_V f_i^V u_i \, dV - \int_S f_i^S u_i \, dS$$

With ε_{ij} : mechanical strain resulting of the loads
σ_{ij}^0 : initial stress state (thermal or mechanical pre-stress).
C_{ijkl} : elastic tensor
f_i^V : vector of forces applied on the volume of the structure
f_i^S : vector of forces applied to the surface of the structure

The deformations are then computed from the mathematical derivations of the displacements.

It could be defined that the equilibrium of structure is obtained when the total potential energy of the structure is minimal. From this condition, it could be defined the stationary principle or principle of minimum total potential energy.
Based on this structural equilibrium, it can be observed that whatever the **perturbation of displacement (virtual displacement)** that **do not modify the value of total potential energy**.

$$\delta u_i \neq 0 \quad \Rightarrow \quad \delta \Pi = 0$$

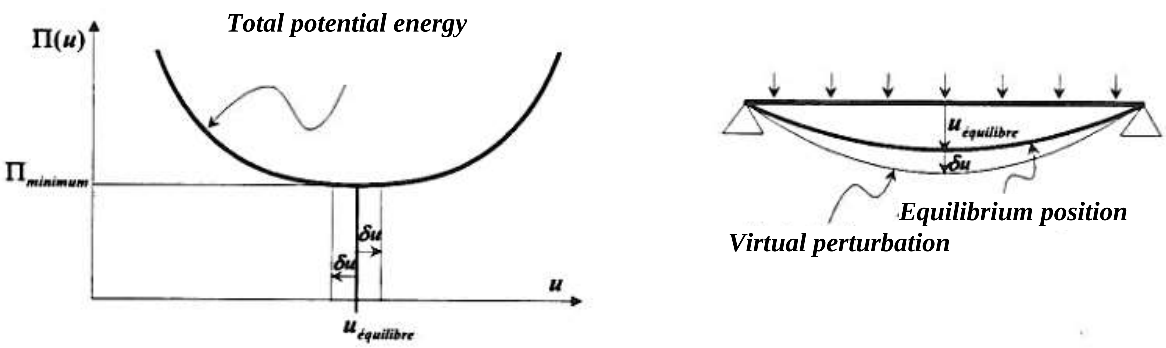
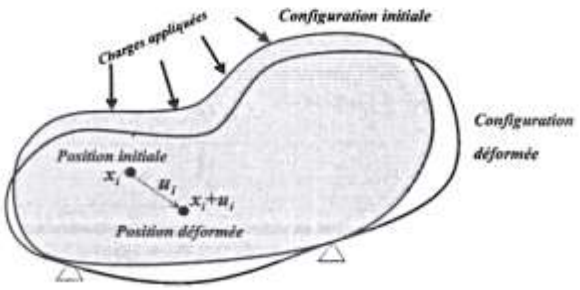


Figure 6 : illustration of the principle of a minimum total potential energy and virtual perturbation.

7.1 Formulation of elementary matrix

Let consider a solid subjected to external loads, these loads will generate deformations on the solid.

7.1.1 Definition of the displacement fields



Let suppose (x, y, z) becomes $(x + u,\ y + v,\ z + w)$

As shown previously, the displacement field of an element is defined by interpolating the nodal values u_i:

$$u\,(x, y, z) = \sum_{i=1}^{n_e} N_i\,(x, y, z)\ u_i$$

$$\{u\} = \begin{bmatrix} u(x, y, z) \\ v(x, y, z) \\ w(x, y, z) \end{bmatrix} = \sum_{i=1}^{n_e} \begin{bmatrix} Ni & 0 & 0 \\ 0 & Ni & 0 \\ 0 & 0 & Ni \end{bmatrix} . \begin{bmatrix} u_i \\ v_i \\ w_i \end{bmatrix}$$

With u_i, v_i, w_i, the nodal displacements

Let define a nodal vector of displacements called $\{q_e\}$ defined as:

$$\{q_e\}^t = \{u_1, v_1, w_1, u_2, v_2, w_2, \dots\ u_n, v_n, w_n\}$$

In the general case, we can write: $\underset{(nc \times 1)}{\{u\}} = \underset{(nc \times ndof)}{[\,N\,]}\ \underset{(ndof \times 1)}{\{q_e\}}$

Where nc is the number of displacement fields and ndof is the number of degrees of freedom.

We can rewrite this relation as follow:

$$\{u\} = \left[\begin{bmatrix} N_1 & 0 & 0 \\ 0 & N_1 & 0 \\ 0 & 0 & N_1 \end{bmatrix} \begin{bmatrix} N_2 & 0 & 0 \\ 0 & N_2 & 0 \\ 0 & 0 & N_2 \end{bmatrix} \dots \begin{bmatrix} N_3 & 0 & 0 \\ 0 & N_3 & 0 \\ 0 & 0 & N_3 \end{bmatrix} \right] .\{q_e\}$$

7.1.2 Set up of deformation fields

Assuming the hypothesis related to linear elasticity, the deformation values are derived from the derivative computation of the displacements:

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Or, in more general relation:

$$\{\varepsilon\} = [D]\ \{u\}$$

With $[D]$: matrix of partial derivative operators

In case of planar elasticity problem, the matrix system can be written as:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{Bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

By replacing {u} with its expression as a function of nodal values, we obtain:

$$\underset{(n\varepsilon \times 1)}{\{\varepsilon\}} = \underset{(n\varepsilon \times nc)}{[D]} \underset{(nc \times ndof)}{[N]} \underset{(ndof \times 1)}{\{q_e\}}$$

With nε is the number of deformation parameters in the studied problem.
It could be deduced:

$$\underset{(n\varepsilon \times 1)}{\{\varepsilon\}} = \underset{(n\varepsilon \times ndof)}{[B]} \underset{(ndof \times 1)}{\{q_e\}}$$

where $[B]$ is the matrix composed of the derivative values of the shape functions.

The matrix $[B]$ is then independent of the displacement state of the element.

7.1.3 Set up of the stress state

The stress values are obtained from the following relationship:

$$\underset{(n\varepsilon \times 1)}{\{\sigma\}} = \underset{(n\varepsilon \times n\varepsilon)}{[C]} \underset{(n\varepsilon \times 1)}{\{\varepsilon\}}$$

$[C]$ matrix of linear elasticity defined by E and ν .

With initial deformation $\{\varepsilon_0\}$ (ex. thermal …), it could be written:

$$\{\sigma\} = [C] (\{\varepsilon\} - \{\varepsilon_0\})$$

By introducing the previous relation and nodal variables:

$$\{\sigma\} = [C]\,[B]\,\{q_e\} - [C]\,\{\varepsilon_0\}$$
$$= [C]\,[B]\,\{q_e\} - \{\sigma_0\}$$

With $\{\sigma_0\}$: initial stress $\{\sigma_0\} = [C]\,\{\varepsilon_0\}$

7.1.4 Summary of the main results

We have shown that the behavior of the structure by finite element procedure is fully defined according to the nodal unknown parameters $\{q_e\}$. To verify the stationary of the total potential energy, a virtual variation must be applied through a perturbation of the displacement field.
A key result is then, all other quantities being independent then the operator (perturbation) is only applied to nodal variables.

$$\{\partial u\} = [N]\,\{\partial q_e\}$$
$$\{\partial \varepsilon\} = [B]\,\{\partial q_e\}$$
$$\{\partial \sigma\} = [C]\,[B]\,\{\partial q_e\}$$

7.2 Construction of the elementary matrix

$$\Pi_e = \int_{V_e} \left(\frac{1}{2} C_{ijkl}\varepsilon_{ij}\varepsilon_{kl} - \sigma_{ij}^0\varepsilon_{ij}\right) dV - \int_{V_e} f_i^V u_i \, dV - \int_{S_e} f_i^S u_i \, dS$$

With Ve and Se represent respectively the volume and the surface of the element

This expression can be written as:

$$\Pi_e = \int_{V_e} \left(\frac{1}{2} \{\varepsilon\}^t[C]\{\varepsilon\} - \{\varepsilon\}^t\{\sigma^0\}\right) dV - \int_{V_e} \{u\}^t\{f^V\} \, dV - \int_{S_e} \{u\}^t\{f^S\} \, dS$$

The virtual variation of {δu} leads to:

$$\delta\Pi_e = \int_{V_e} \left(\{\delta\varepsilon\}^t[C]\{\varepsilon\} - \{\delta\varepsilon\}^t\{\sigma^0\}\right) dV - \int_{V_e} \{\delta u\}^t\{f^V\}\, dV - \int_{S_e} \{\delta u\}^t\{f^S\}\, dS$$

The discretization of this expression gives:

$$\delta\Pi_e = \int_{V_e} \left(\{\delta q_e\}^t[B]^t[C][B]\{q_e\} - \{\delta q_e\}^t[B]^t\{\sigma^0\}\right) dV - \int_{V_e} \{\delta q_e\}^t[N]^t\{f^V\}\, dV - \int_{S_e} \{\delta q_e\}^t[N]^t\{f^S\}\, dS$$

If $\{\delta q_e\}^t$ is put in common in the relation:

$$\delta\Pi_e = \{\delta q_e\}^t\left(\int_{V_e} \left([B]^t[C][B]\{q_e\} - [B]^t\{\sigma^0\}\right) dV - \int_{V_e} [N]^t\{f^V\}\, dV - \int_{S_e} [N]^t\{f^S\}\, dS\right)$$

However, by using the stationary of the potential energy, this equation could be simplified (whatever the value of $\{\delta q_e\}^t$):

$$\underbrace{\left(\int_{V_e} [B]^t[C][B]\; dV\right)}_{\text{Stiffness of the structure: Material, geometry, contacts}} \{q_e\} = \underbrace{\int_{V_e} [B]^t\{\sigma^0\} dV + \int_{V_e} [N]^t\{f^V\}\, dV + \int_{S_e} [N]^t\{f^S\}\, dS}_{\text{Solicitation of the structure : Volume or surface loads, initial stress state}}$$

From this relation, it could be written the following relation:

$$[K_e] \qquad \{q_e\} \;=\; \{F_e\}$$

$$\left(\int_{V_e} [B]^t[C][B]\; dV\right) \{q_e\} = \int_{V_e} [B]^t\{\sigma^0\} dV + \int_{V_e} [N]^t\{f^V\}\, dV + \int_{S_e} [N]^t\{f^S\}\, dS$$

It could be deduced by identification:

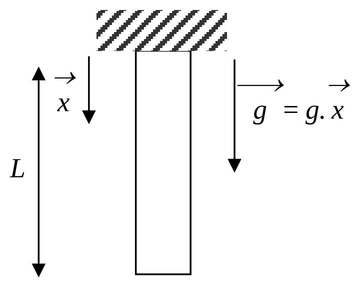
$$[K_e] = \left(\int_{V_e} [B]^t[C][B]\; dV\right) : \textbf{Stiffness matrix}$$

And : $\{F_e\} = \{F_e^V\} + \{F_e^S\} + \{F_e^0\}$

$$\{F_e^V\} = \int_{V_e} [N]^t\{f^V\}\, dV \;: \textbf{Vector of volume loads}$$

$$\{F_e^S\} = \int_{S_e} [N]^t\{f^S\}\, dS : \textbf{Vector of surface loads}$$

7.3 Illustration on a weighted bar



Let supposed a bar subjected to its own weight.

1. Carry out the theoretical study of the bar fields (deformations, stresses, displacements)
2. Make an approximation with the use of:
   - 1 truss element
   - 2 truss elements and finally 3 trusses

Analytical approach

The normal load applied in the section is: F(x) = $\rho g S(L - x)$

The relation between the load and the displacement following the Bernouilli's hypothesis:

$$\frac{du(x)}{dx} = \frac{F(x)}{ES} = \frac{\rho g}{E}(L - x)$$

Then: $$u(x) = \int \frac{\rho g}{E}(L - x)dx = \frac{\rho g}{E}\int (L - x)dx = \frac{\rho g}{2E}x(2L - x)$$

Finite element approach

The stiffness matrix can be defined by:

$$[K_e] = \int_{V_e} [B]^t[C][B]\ dV$$

And for 1 element,

$$\varepsilon(x) = \frac{du(x)}{dx} = \frac{dN1(x)}{dx}u_1 + \frac{dN2(x)}{dx}u_2 = \left[\frac{dN1(x)}{dx}\ \ \frac{dN2(x)}{dx}\right]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

$$\sigma(x) = E\ .\ \varepsilon(x)$$

We can deduce by identification the different terms:

$$[B] = \left[\frac{dN1(x)}{dx}\ \ \frac{dN2(x)}{dx}\right]$$

$$[C] = E$$

For a bar with a constant section value,

$$[K_e] = \int_l \int_S \begin{bmatrix} \frac{dN1(x)}{dx} \\ \frac{dN2(x)}{dx} \end{bmatrix} E \left[\frac{dN1(x)}{dx}\ \ \frac{dN2(x)}{dx}\right] dx\ dS$$

$$[K_e] = ES\int_l \begin{bmatrix} \frac{dN1(x)}{dx}\frac{dN1(x)}{dx} & \frac{dN1(x)}{dx}\frac{dN2(x)}{dx} \\ \frac{dN2(x)}{dx}\frac{dN1(x)}{dx} & \frac{dN2(x)}{dx}\frac{dN2(x)}{dx} \end{bmatrix} dx$$

With $N1(x) = 1 - \frac{x}{l} \rightarrow \frac{dN1(x)}{dx} = -\frac{1}{l}$

$N2(x) = \frac{x}{l} \rightarrow \frac{dN2(x)}{dx} = \frac{1}{l}$

We can find that:

$$[B] = \left[-\frac{1}{l}\ \ \frac{1}{l}\right]$$

And the stiffness matrix corresponds to:

$$[K_e] = \frac{ES}{l}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

For the computation of the load due to the own weight of the bar, the equivalent nodal loads can be calculated by:

$$\{F_e\} = \int_{V_e} [N]^t\{f^v\}dV = S\int_l [N]^t\{f^v\}dx$$

With $\{f^v\} = \rho g$

$$\{F_e\} = S\int_0^l \begin{Bmatrix} 1-\frac{x}{l} \\ \frac{x}{l} \end{Bmatrix} \rho g\, dx = \rho g S l \begin{Bmatrix} \frac{1}{2} \\ \frac{1}{2} \end{Bmatrix}$$

With $\rho g S l$ corresponds to the total weight of the bar, called later W.

For 1 element:

$$\frac{ES}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = W\begin{Bmatrix} \frac{1}{2} \\ \frac{1}{2} \end{Bmatrix}$$

So $\quad u_2 = \dfrac{WL}{2ES}$

$$\varepsilon(x) = \frac{-1}{L}u_1 + \frac{1}{L}u_2 = \frac{W}{2ES}$$

$$\sigma(x) = \frac{W}{2S}$$

For 2 elements: and l = L/2 , u_1 = 0

$$\frac{2ES}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{W}{2}\begin{Bmatrix} \frac{1}{2} \\ 1 \\ \frac{1}{2} \end{Bmatrix}$$

So, the solution is:

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{WL}{8ES}\begin{Bmatrix} 0 \\ 3 \\ 4 \end{Bmatrix}$$

For x < L/2 : $\varepsilon^1 = \dfrac{-1}{L/2}u_1 + \dfrac{1}{L/2}u_2 = \dfrac{3W}{4ES} \rightarrow \sigma^1 = \dfrac{3W}{4S}$

For x > L/2 : $\varepsilon^2 = \dfrac{-1}{L/2}u_2 + \dfrac{1}{L/2}u_3 = \dfrac{W}{4ES} \rightarrow \sigma^2 = \dfrac{W}{4S}$

For 3 elements: l = L/3 , u_1 = 0

$$\frac{3ES}{L}\begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \frac{W}{3}\begin{Bmatrix} \frac{1}{2} \\ 1 \\ 1 \\ \frac{1}{2} \end{Bmatrix}$$

So, the solution is:

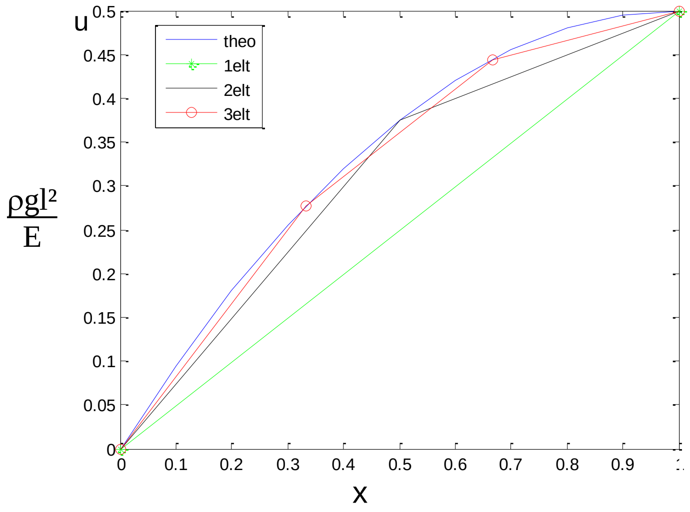
$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \frac{WL}{18ES}\begin{Bmatrix} 0 \\ 5 \\ 8 \\ 9 \end{Bmatrix}$$

For x < L/3 : $\varepsilon^1 = \dfrac{-1}{L/3}u_1 + \dfrac{1}{L/3}u_2 = \dfrac{5W}{6ES} \rightarrow \sigma^1 = \dfrac{5W}{6S}$

For L/3 < x < 2L/3 : $\varepsilon^2 = \dfrac{-1}{L/3}u_2 + \dfrac{1}{L/3}u_3 = \dfrac{W}{2ES} \rightarrow \sigma^2 = \dfrac{W}{2S}$

For 2L/3 < x < L : $\varepsilon^3 = \dfrac{-1}{L/3}u_3 + \dfrac{1}{L/3}u_4 = \dfrac{W}{6ES} \rightarrow \sigma^3 = \dfrac{W}{6S}$

7.3.1 Synthesis of the main results and conclusion



We can see the comparison between analytical solution and finite element solutions.

Even if the nodal displacement values are equal to the analytical results, the shape of the displacement is not exact.

Some errors appear!

The increase of the element numbers leads to decrease these errors.

8 Exercises and Practical Work Support

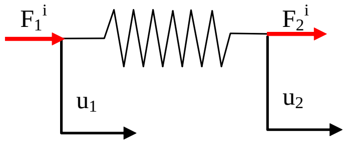
8.1 The bar case

8.1.1 Exercice 1:

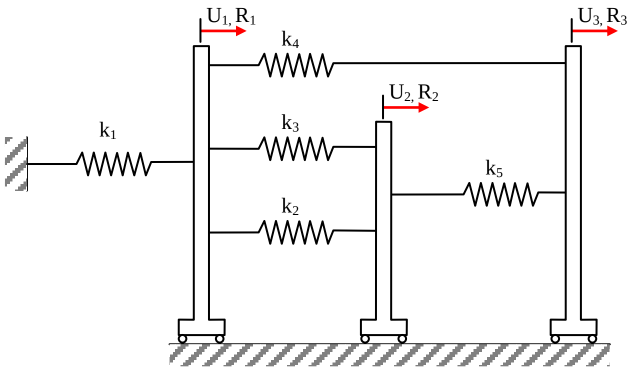
The figure below shows a mechanical system consisting of 3 rigid mobile trolleys interconnected by a system of stiffness springs called k^i.

Q1. Give the equilibrium expression of particular spring k^i subjected to external loads F_1^i et F_2^i and nodal displacements u_1 et u_2. Deduce the relationship {F}=[K]{u} following two different approaches:

- By writing the equilibrium expression for every node.
- By using Castigliano's theorem (every derive of the potential energy against every displacement is null).

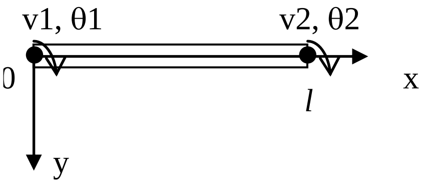


Q2. Calculate the displacement u_i of every rigid mobile and the forces F_i in the springs for the applied loading R_i.



8.2 Definition of a finite element for the bending: beam element

8.2.1 Bending behavior



A beam element is a 2 node element of class C1 (ensuring the tangent continuity).
We therefore want to write the displacement according to y (called v) as follows:
$v(x) = N_1(x).v_1 + N_2(x).\theta_1 + N_3(x).v_2 + N_4(x).\theta_2$

The following conditions are defined:

$v(0) = v_1 \qquad N_1 = v_1$ et $N_2 = N_3 = N_4 = 0$
$v'(0) = \theta_1 \qquad N'_2 = \theta_1$ et $N_1' = N_3' = N_4' = 0$
$v(l) = v_2 \qquad N_3 = v_2$ et $N_1 = N_2 = N_4 = 0$
$v'(l) = \theta_2 \qquad N_4 = \theta_2$ et $N_1' = N_2' = N_3' = 0$

In order to respect all these conditions, the polynomial must be of the 3rd order.
Then, $\qquad N_i(x) = a + bx + cx^2 + dx^3$
and $\qquad N_i'(x) = b + 2cx + 3dx^2$

The following system can be built:

$$\begin{vmatrix} v(0) = v_1 \\ v'(0) = \theta_1 \\ v(l) = v_2 \\ v'(l) = \theta_2 \end{vmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 1 & l & l^2 & l^3 \\ 0 & 1 & 2l & 3l^2 \end{bmatrix} . \begin{vmatrix} a \\ b \\ c \\ d \end{vmatrix}$$

By solving this system, we can obtain:

$$\begin{vmatrix} a \\ b \\ c \\ d \end{vmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -3/l^2 & -2/l & 3/l^2 & -1/l \\ -2/l^3 & 1/l^2 & -2/l^3 & 1/l^2 \end{bmatrix} . \begin{vmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{vmatrix}$$

It can be seen that each of the columns corresponds to the shape functions N_i, hence :

$N_1(x) = 1 - 3/l^2 .x^2 + 2/l^3\, x^3$
$N_2(x) = x - 2/l .x^2 + 1/l^3\, x^3$
$N_3(x) = 3/l^2 .x^2 - 2/l^3\, x^3$
$N_4(x) = - 1/l .x^2 + 1/l^2\, x^3$

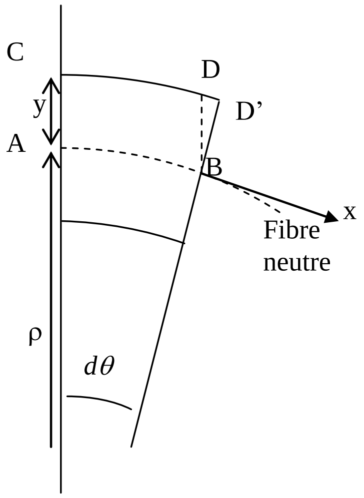
We have seen in the course session that from the shape functions (and their derivatives), it was possible to deduce the stiffness matrix of the element. To do this, it is necessary to express the link between the positions (v_i, θ_i) and the deformations.

8.3 The structural behavior of a structure subject to bending

Hypothesis:

No deformation of the middle of the part
No rotation of the cross section (hyp. Bernoulli)
Linear material behavior

8.3.1 Definition of the geometrical relations for deformations

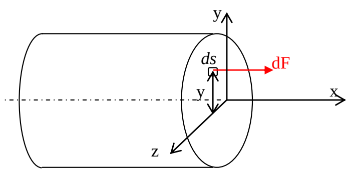


Let suppose a beam subject to a pure bending solicitation (only one torque) around the perpendicular axis to the beam. It results an angular deviation called $d\theta$.
Under the hypotheses, it is assumed that there is no deformation at the middle of the part (neutral fiber), thus:
$AB = \rho d\theta$
On the upper side of the part, it is observed an elongation of the part (intially called CD with deformation) becoming CD' after elongation. It could be deduced:
$CD' = (\rho + y)d\theta$
And finally, the deformation:

$$\varepsilon = \frac{\Delta l}{l_0} = \frac{CD' - CD}{CD} = \frac{DD'}{CD} = \frac{(\rho+y)d\theta - \rho d\theta}{\rho d\theta} = \frac{y}{\rho}$$

By observing that $1/\rho = \frac{d^2v(x)}{dx^2}$ (2nd derivatives of the position along y axis), we can obtained : $\varepsilon = y.\frac{d^2v(x)}{dx^2}$.

8.3.2 Definition of the relation between loads and deformations



Assuming a Hooke law, it could be defined a relation between the stress state and the deformation:

$$\sigma = E\,\varepsilon = E\,\frac{y}{\rho}$$

From these relations, it is possible to write the relation between a local load (dF) and the stress value:

$$dF = \sigma ds = E\,\frac{y}{\rho}\,ds$$

8.4 *Stiffness matrix of a beam*

From the relation defined in 2.1: $\varepsilon = \frac{y}{\rho} = y.\frac{d^2v(x)}{dx^2}$. It could be writen for a discrete problem formulation:

$$\varepsilon(x) = y\left[\frac{d^2N_1(x)}{dx^2} \;\ldots\; \frac{d^2N_4(x)}{dx^2}\right].\begin{vmatrix} v_1 \\ \ldots \\ \theta_2 \end{vmatrix}$$

The corresponding derivatives relations:

$$\frac{d^2N_1(x)}{dx^2} = -6/l^2 + 12x/l^3 \qquad \frac{d^2N_2(x)}{dx^2} = -4/l + 6x/l^2$$

$$\frac{d^2N_3(x)}{dx^2} = 6/l^2 - 12x/l^3 \qquad \frac{d^2N_4(x)}{dx^2} = -2/l + 6x/l^2$$

Then,

$$[K_e] = \int_l \int_S \begin{bmatrix} \frac{d^2N1(x)}{dx^2} \\ \dots \\ \frac{d^2N4(x)}{dx^2} \end{bmatrix} y^2 E \left[\frac{d^2N1(x)}{dx^2} \ \dots \ \frac{d^2N4(x)}{dx^2} \right] dx\, dS$$

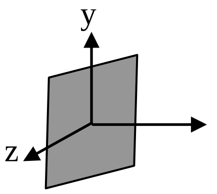
A finally:

$$[K_e] = \int_l \begin{bmatrix} \frac{d^2N1(x)}{dx^2} \\ \dots \\ \frac{d^2N4(x)}{dx^2} \end{bmatrix} E \left[\frac{d^2N1(x)}{dx^2} \ \dots \ \frac{d^2N4(x)}{dx^2} \right] dx \int_S y^2\, dS$$

Thus, it could identify the definition of the area moment of inertia, $\int_S y^2\, dS$, it could deduced:

$$\int_S y^2\, dS = \int_y \int_z y^2\, dy\, dz$$

$$= \left[\frac{1}{3} y^3 \right]_{-h/2}^{h/2} \left[z \right]_{-b/2}^{b/2} = \frac{bh^3}{12}$$



$z \in [-b/2 ; b/2]$
$y \in [-h/2 ; h/2]$

The final integration of this relation gives the symmetric stiffness matrix $[K_e]$:

$$[K_e] = \frac{EI}{l^3} \cdot \begin{bmatrix} 12 & 6l & -12 & 6l \\ & 4l^2 & -6l & 2l^2 \\ & & 12 & -6l \\ & & & 4l^2 \end{bmatrix}$$

8.4.1 Global behavior of beam (including traction and bending)

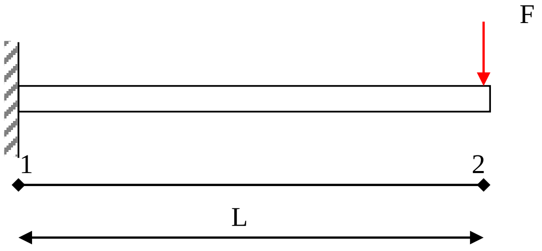
The element has a similar behavior of a bar for tensile / compressive loads and a bending behavior, which has been explained in the previous paragraph. We can deduce the global stiffness matrix:

$$[K_e] = \begin{array}{c} \begin{array}{cccccc} u1 & v1 & \theta 1 & u2 & v2 & \theta 2 \end{array} \\ \left| \begin{array}{cccccc} ES/l & 0 & 0 & -ES/l & 0 & 0 \\ & 12EI/l^3 & 6EI/l^2 & 0 & -12EI/l^3 & 6EI/l^2 \\ & & 4EI/l & 0 & -6EI/l^2 & 2EI/l \\ & & & ES/l & 0 & 0 \\ & & & & 12EI/l^3 & -6EI/l^2 \\ & & & & & 4EI/l \end{array} \right| \end{array} \begin{array}{c} \\ u1 \\ v1 \\ \theta 1 \\ u2 \\ v2 \\ \theta 2 \end{array}$$

8.5 *First application*

8.5.1 Concentrated force case

Data from the practical work application:



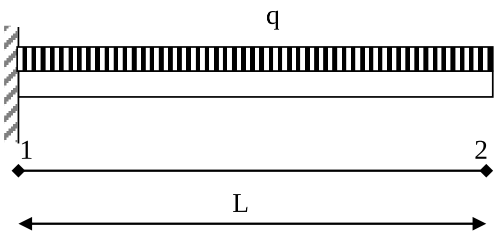
width b=0.1m ;
L=4m ;
F=2000N ;
h= 0.15m

It is proposed to model this application with only **1 beam element**. Following these different steps:

1. Definition of the elementary stiffness matrix
2. Definition of the global stiffness matrix
3. Introduction of the boundary conditions
4. Resolution

8.5.2 Linear load case

It is proposed to study the same application assuming a linear load on the structure (q).



Width b=0.1m ;
L=4m ;
q=500N/m
h= 0.15m

Definition of the equivalent nodal load to be applied:

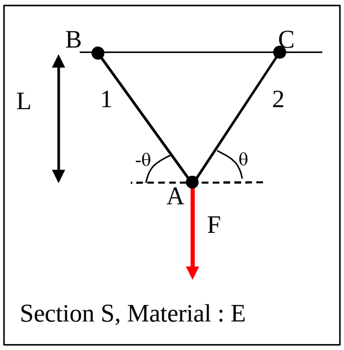
$$\{Fe\}=\int_{V_e}[N]^t\{f^v\}dV=\int_{l}[N]^t\{f^v\}dx \text{ with } \{f^v\}=q$$

1. Computation of the equivalent nodal load to be applied
2. Definition of the elementary stiffness matrix
3. Definition of the global stiffness matrix
4. Introduction of the boundary conditions
5. Resolution

8.6 *From a local to a global stiffness behavior*

8.6.1 Studied case

Let supposed an articulated lattice made up of bar elements. Each of them can then be subjected to traction or compression solicitations.



8.6.2 Discrete modeling

The finite element approach can be composed of:

- 2 elements
- 3 nodes

Thus, the table could be deduced:

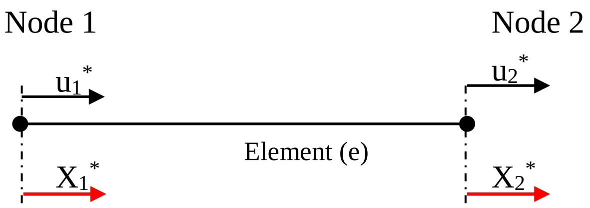
	Node	Section	Length
1	*A,B*	*S*	*L/sin θ*
2	*A,C*	*S*	*L/sin θ*

The vectors of nodal displacements and forces are then:

$$\{q\}=\begin{Bmatrix} u_A \\ v_A \\ u_B \\ v_B \\ u_C \\ v_C \end{Bmatrix} \quad \text{and} \quad \{F\}=\begin{Bmatrix} Fx_A \\ Fy_A \\ Fx_B \\ Fy_B \\ Fx_C \\ Fy_C \end{Bmatrix}$$

8.6.3 The definition of the truss element

Description



In the local frame of the truss, the nodal displacements and forces are along the x axis

$$\{q_e\}=\begin{Bmatrix} u_1^* \\ u_2^* \end{Bmatrix}$$

and $$\{F_e\}=\begin{Bmatrix} X_1^* \\ X_2^* \end{Bmatrix}$$

Interpolation

The displacement and the shape function are:

$$U_e^*(x) = u_1^* \left(1 - \frac{x}{L}\right) + u_2^* \frac{x}{L} \qquad \begin{cases} N_1(x) = \left(1 - \frac{x}{L}\right) \\ N_2(x) = \frac{x}{L} \end{cases}$$

Then:

$$\begin{aligned} U_e^*(x) &= N_1(x) \,.\, u_1^* + N_2(x) \,.\, u_2^* \\ &= [N_1(x) \quad N_2(x)] \,.\, \begin{Bmatrix} u_1^* \\ u_2^* \end{Bmatrix} \\ &= [N_e]\,\{q_e\} \end{aligned}$$

Behavior material and stiffness matrix

Hooke law is defined as $\sigma_{11}=E\varepsilon_{11}$; and finally, [C]=[E]

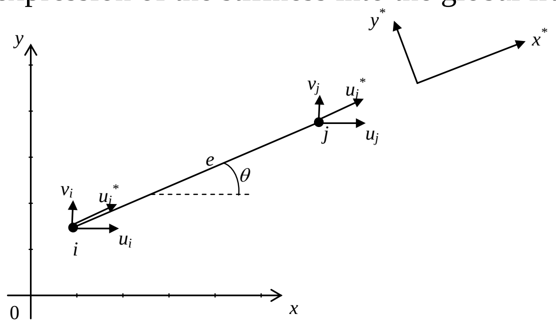
Thus,

$$[K_e^*] = \int_{V_e} [B_e]^t[C][B_e]\ dV \quad \text{such as } [B_e] = [-\frac{1}{L} \quad \frac{1}{L}]$$

And finally $$[K_e^*] = ES_e \int_l \begin{bmatrix} \frac{1}{L^2} & \frac{-1}{L^2} \\ \frac{-1}{L^2} & \frac{1}{L^2} \end{bmatrix} dx = \frac{\mathbf{ES_e}}{\mathbf{L_e}} \begin{bmatrix} \mathbf{1} & \mathbf{-1} \\ \mathbf{-1} & \mathbf{1} \end{bmatrix}$$

8.6.4 From the local to global reference frame

The previous stiffness matrix is defined in the local frame. The rotation leads to modify the expression of the stiffness into the global frame of the structure.



Let supposed $\{q_e\}$ and $\{F_e\}$ be the nodal displacement and load on the element (e) expressed in the global frame.

Let supposed $\{q_e^*\}$ and $\{F_e^*\}$ be the nodal displacement and load on the element (e) expressed in the local frame (x^*, y^*).

It is possible to express the movements of the nodes in the local frame, according to the movements in the global frame:

$$\begin{cases} u_i^* = u_i \cos\theta + v_i \sin\theta \\ u_j^* = u_j \cos\theta + v_j \sin\theta \end{cases}$$

We can then express a matrix relationship between the movements in the local frame and the movements in the global frame:

$$\begin{Bmatrix} u_i^* \\ u_j^* \end{Bmatrix}^e = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}^e$$

$$\{q_e^*\} = [\lambda_e] \quad \{q_e\}$$

And then: $$\{F_e^*\} = [\lambda_e] \quad \{F_e\}$$

However, we know that:

$$\{F_e^*\} = [K_e^*]\{q_e^*\}$$
$$\{F_e^*\} = [\lambda_e]\,\{F_e\} = [K_e^*]\,[\lambda_e]\{q_e\}$$

Using the orthogonality properties of the transformation $[\lambda_e]^T[\lambda_e]=[I]$ and multiplying by $[\lambda_e]^T$, it comes:

$$\{F_e\} = [\lambda_e]^T[K_e^*]\,[\lambda_e]\{q_e\}$$

with $\{F_e\} = [Ke]\,\{q_e\}$

By identification:

$$[Ke] = [\lambda_e]^T[K_e^*]\,[\lambda_e]$$

The stiffness matrix in the structural reference frame is deduced from this:

$$[Ke] = \begin{bmatrix} c & 0 \\ s & 0 \\ 0 & c \\ 0 & s \end{bmatrix} \frac{ES}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}.\begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix} \qquad \text{Avec} \begin{cases} c = \cos\theta \\ s = \sin\theta \end{cases}$$

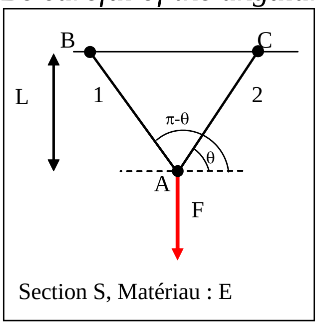
Thus:

$$[Ke] = \frac{ES}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}$$

8.6.5 Back to the application

In the previous lattice assembly, the stiffness matrix is defined by:

Be careful of the angular orientation θ:



Knowing that: $\cos(\pi-\theta) = -\cos(\theta)$
and $\sin(\pi-\theta) = \sin(\theta)$

Truss AB :

$$[K_e^1] = \frac{ES}{L/s}\begin{array}{c} \begin{array}{cccc} u_A & v_A & u_B & v_B \end{array} \\ \begin{bmatrix} c^2 & -cs & -c^2 & cs \\ -cs & s^2 & cs & -s^2 \\ -c^2 & cs & c^2 & -cs \\ cs & -s^2 & -cs & s^2 \end{bmatrix} \end{array}$$

Truss AC :

$$[K_e^2] = \frac{ES}{L/s}\begin{array}{c} \begin{array}{cccc} u_A & v_A & u_C & v_C \end{array} \\ \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \end{array}$$

with $\begin{cases} c = \cos(\theta) \\ s = \sin(\theta) \end{cases}$

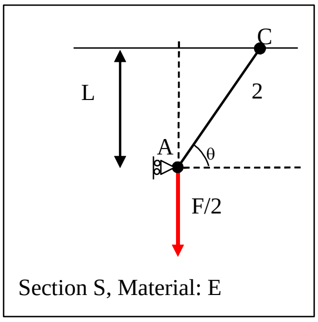
The assembly method requires that all components of the elementary matrices applying to the same degree of freedom be summed up. For the lattice, this leads to the following overall stiffness matrix:

$$[K] = \frac{ES.s}{L}\begin{array}{c} \begin{matrix} u_A & v_A & u_B & v_B & u_C & v_C \end{matrix} \\ \begin{bmatrix} c^2+c^2 & 0 & -c^2 & cs & -c^2 & -cs \\ 0 & s^2+s^2 & cs & -s^2 & -cs & -s^2 \\ -c^2 & cs & c^2 & -cs & 0 & 0 \\ cs & -s^2 & -cs & s^2 & 0 & 0 \\ -c^2 & -cs & 0 & 0 & c^2 & cs \\ -cs & -s^2 & 0 & 0 & cs & s^2 \end{bmatrix} \end{array}$$

8.6.6 Exercise 1

1. Definition of the boundary condition
2. Solving of the problem

8.6.7 Exercise 2: exploitation of the symmetrical condition



Assuming that the problem is symmetric, it is proposed to study this equivalent problem formulation:

Follows these steps:

1. Definition of the assembly stiffness matrix
2. Definition of the boundary condition
3. Solving of the problem

9 Abaqus a finite element software

The main objective of finite element software is to simulate the physical response of structures and solids to external solicitations like mechanical forces, temperatures, impacts and other external conditions.
It is necessary to keep a “physical sense” of the simulations that you will have to perform. Indeed, the specificity of finite element software is to draw result through maps (very demonstrative information) but without any guarantee concerning the relevance of physical meaning. It is very important to know or predict the order of the quantities that you will expect and then, you will be able to analyze the truth of the computed results. Most of the time, complex structure and analyses could be simplified into a simple case for which analytical results are well known (e.g. from continuum mechanics).

9.1 Computed modules of Abaqus

There are two main modules available in the Abaqus software:
- Standard Abaqus: following an implicit integration scheme (imposes the time increment, and the equilibrium state of the forces is deduced from it). That allows linear and non-linear problem solving, with 1D, 2D, Axisymmetric or 3D geometries. Many analysis procedures are available including time or frequency domain.
- Explicit computation: Allows non-linear, transient and dynamic analyses of structures (e.g. Crash Test...). This explicit method of time integration still allows quasi-static studies with important non-linear behaviors.

9.2 Added modules

9.2.1 Abaqus/CAE : graphic environment

The main interface for users of Abaqus is called Abaqus CAE; this graphical interface allows the creation of models, the runs of an analysis and the processing of results.

9.2.2 Abaqus/Post : post traitement

This graphical interface only allows the display of the **results of an analysis such as** isocontours, graphs,...

9.2.3 Other modules :

Abaqus/Design : configuration of the Abaqus models, Sensitivity analysis
Abaqus/Safe : lifetime of a structure.

9.3 CAD interface

Compatibilities are possible with CAD software like Catia, Ideas, Creo... The use of neutral CAD format is another way to import complex geometry into Abaqus (Step, Iges…).

9.4 Global use of Abaqus

The Abaqus software is a solver that solves a problem described in a typical input file where data and sequences of the problem is fully defined. The resolution is written in an "output" file or result file.

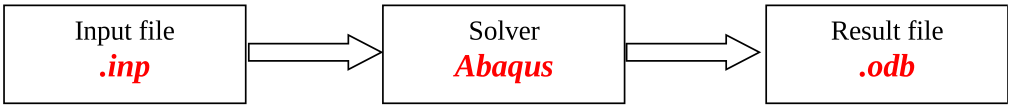


Figure 7 : Main steps of the computation into Abaqus

9.4.1 Input file

Extension : ***.inp***
Content: key words describing the geometry, the material properties, the boundary conditions…

9.4.2 Result file

Extension : ***.odb***
Content : contours, curves, result tables (compiled file)

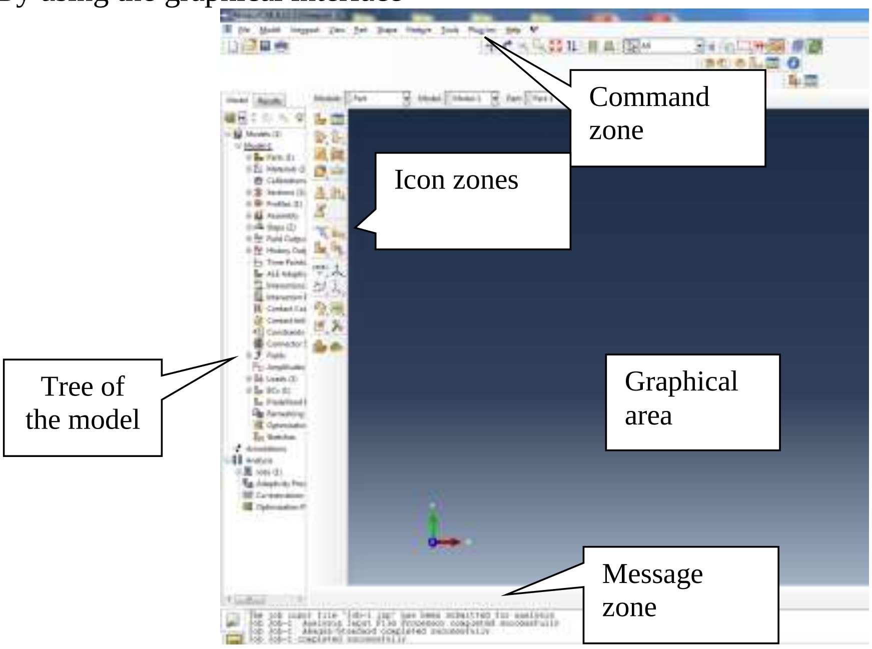
9.4.3 Other files

File ***.com*** :run sequence of the computation
File ***.dat*** : modeling summary file, also containing messages, calculation time..
File ***.msg*** : Summary file of the current calculation, list of error messages during the calculation process.

9.5 How to make a simulation

2 different ways are possible:

- By creating your own input file using a text editor and knowing the different keywords (Abaqus Command and Abaqus viewer help and, of course, help)
- By using the graphical interface



To set up a simulation under Abaqus it requires to follow successively in the different modules:

1. Part
2. Property
3. Assembly
4. Step
5. Interaction

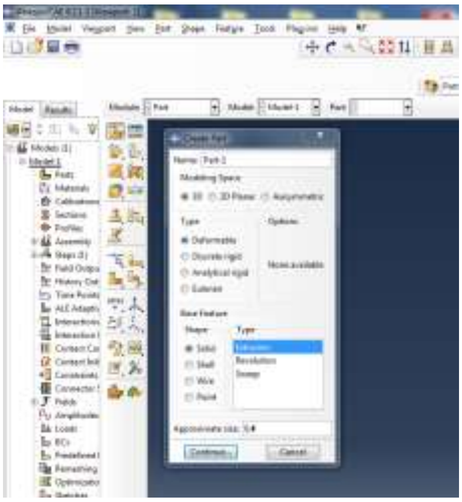
6. Load
7. Mesh
8. Job
9. Visualization (to see the results)

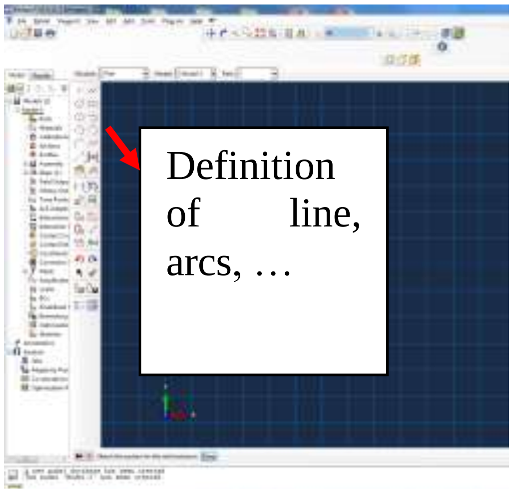
9.6 *The main modules of Abaqus*

9.6.1 Part

Creation of the structural parts of the simulation to be performed.

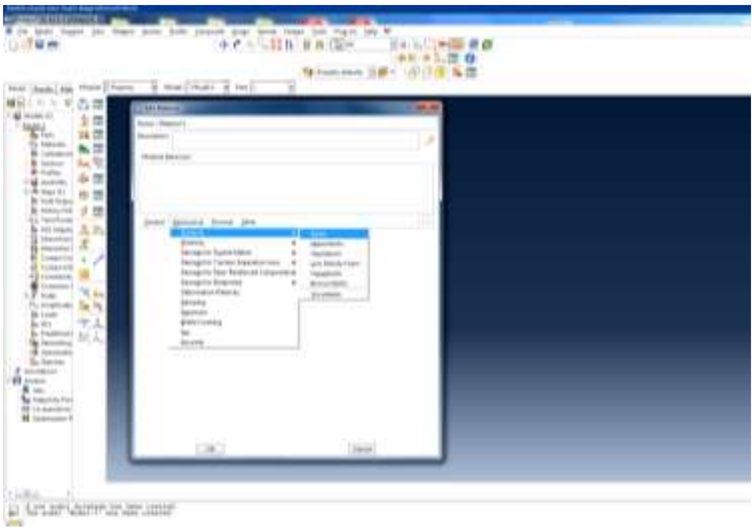
- - These drawings can be directly created in Abaqus with the CAD modeler
- - Or import (.sat,.iges,.stp,.prt,.wrml, ...) from other CAD software.





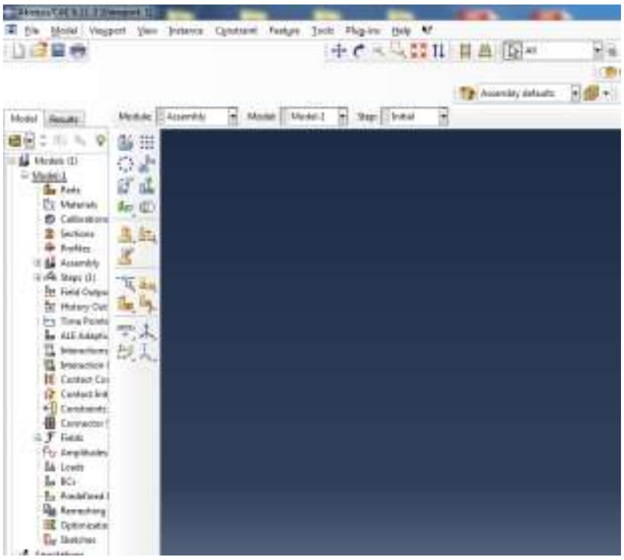
9.6.2 Property module

Allows to create materials, assign sections, orientation markers (mainly for beams or shells), shape of profiles....



9.6.3 Assembly Module

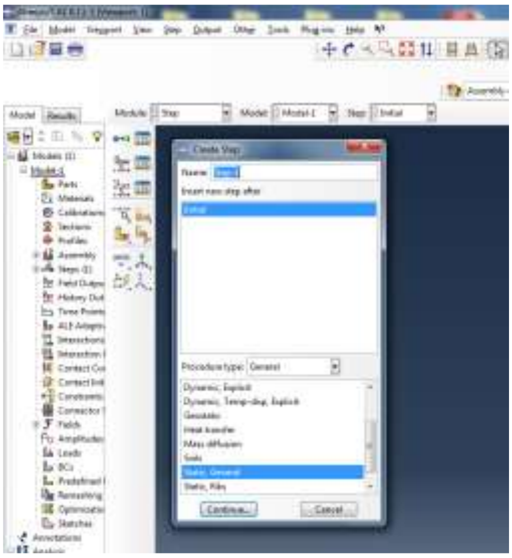
This module allows to insert parts, to position them between others by geometric operations (translations, rotations) or with contact constraints



9.6.4 Step Module

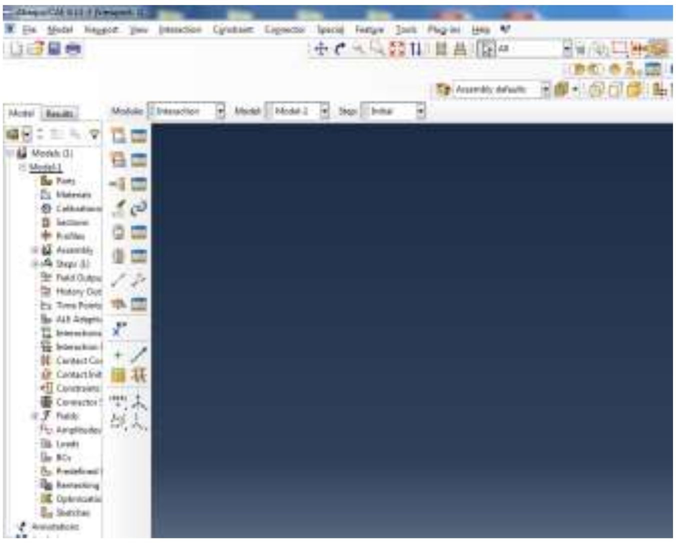
Module for defining the type of calculation that will be performed (static, dynamic...)

It is also necessary to choose the calculation variables that will be available in the exploitation of results and output data (History Output and Field Output).



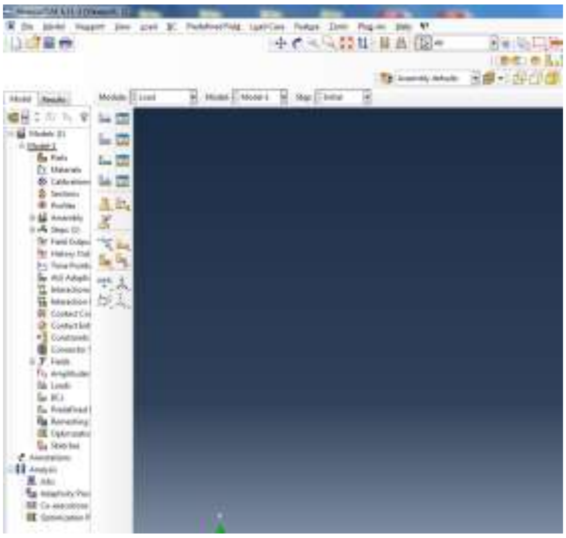
9.6.5 Interaction Module

Module allowing to create interactions between parts, defines contact properties, create constraints, connectors (links)...



9.6.6 Load Module

Definition of forces and boundary conditions, fields and loading conditions (temperature, pressure...)

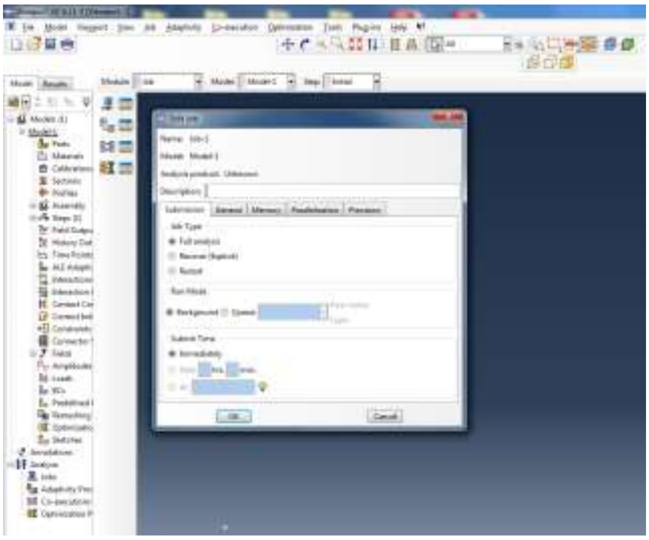


9.6.7 Mesh Module

Creation of the partitions of the different entities, assigning the control of the mesh, choice of elements, mesh and verification tools.

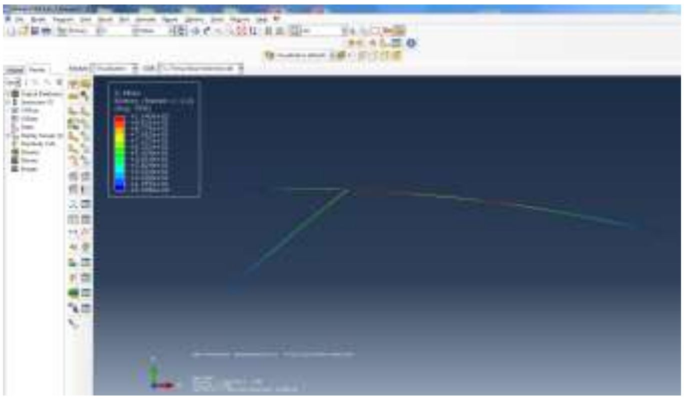
9.6.8 Job Module

Creation of the calculation file, selection of the accuracy, number of processors used....



9.6.9 Visualization Module

Display of computed displacements of the structure, modes with results either in graphical form or as vectors. Animations during loading, choice of color palettes, creation of output fields with calculation options on the fields are also possible....



9.7 *First example*

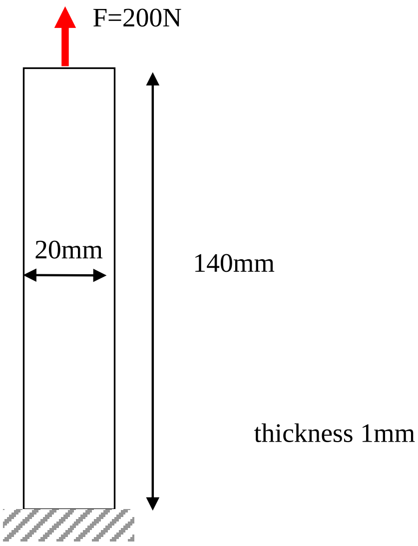
The purpose of this first example is to perform a tensile test. For this, we will realize 4 different models of this operation. Either by using:

- truss elements,
- beam elements
- shell elements

and finally, solid elements.

The tensile test is a test whose stresses are located in a plane, the plane of the specimen. This test is a standardized test from which the main characteristics of the material are derived.

9.7.1 Conditions of the test



The material used is an aluminum alloy. The plastic behavior is linear, with:
E = 80 000MPa
ν=0.3
Density = 2700kg/m^3

Steady state simulation.

The part is blocked into the tensile test machine.

Attention : verify if the units of the model are homogeneous

Static (i.e. abaqus standard formulation), you can work either in millimeters or meter

In dynamics, work only in units of the International System (m).

9.7.2 Analytical analysis of the problem

In order to quickly determine the expected forces, stresses and deformations, it is necessary to make a simple model of the bar in tension. **Deduce the main quantities from this**. These values can be directly compared to the computed values by Abaqus.

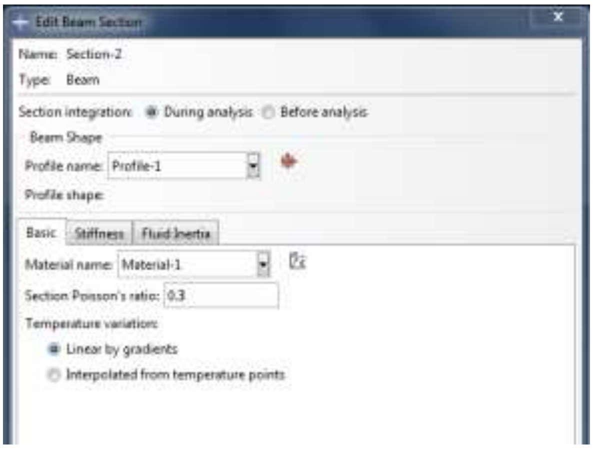
9.7.3 Truss / beam model

Be careful, it is advisable to regularly save your work all along the progress of the practical work !

Create a new file called « Traction_Beam »

Module part :
Create part *2D planar*, *deformable*, option *Wire*. Design the length of the line corresponding to the part.

Module material
Create a material designated by « alu », in the *general* menu, define the *density*.
Then, go to *Mechanical* menu and define the elastic behavior of the material (*Elastic*).
Go to *section* menu, select *Beam*, then, create a profile, *rectangular* and define the dimensions of the part section (pay attention to the orientation of the axis). Attribute the material to the part.



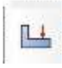
Assembly Module
Import the created beam and position it to the origin of the frame.

Step Module
Create a computed step, call it "traction", select *Static*, *general*

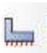
Have a look of the computed value during the simulation *Fields output* and *History Output*.

Interaction Module
A priori, during this particular test, there is no interaction between different parts. Therefore, there is nothing to report.

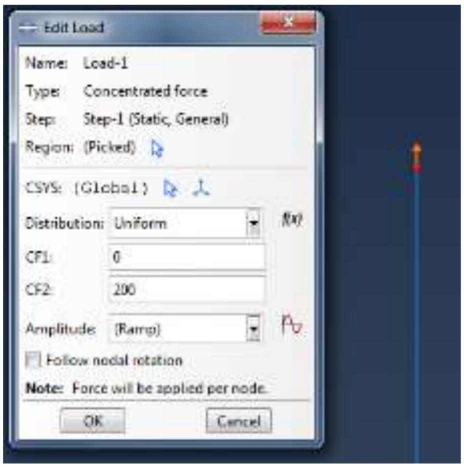
Load Module



Create a Force type *load* applied to the end of the beam.



Declare the embedding conditions at the other end of the specimen



Mesh Module

You should select the element type B21 or 31, verify in the horizontal menu *Mesh* → *Element type*.

Then, you should create *seeds* to determine the size of the mesh and then to generate the mesh.

In this case, use 10 elements.

Job Module

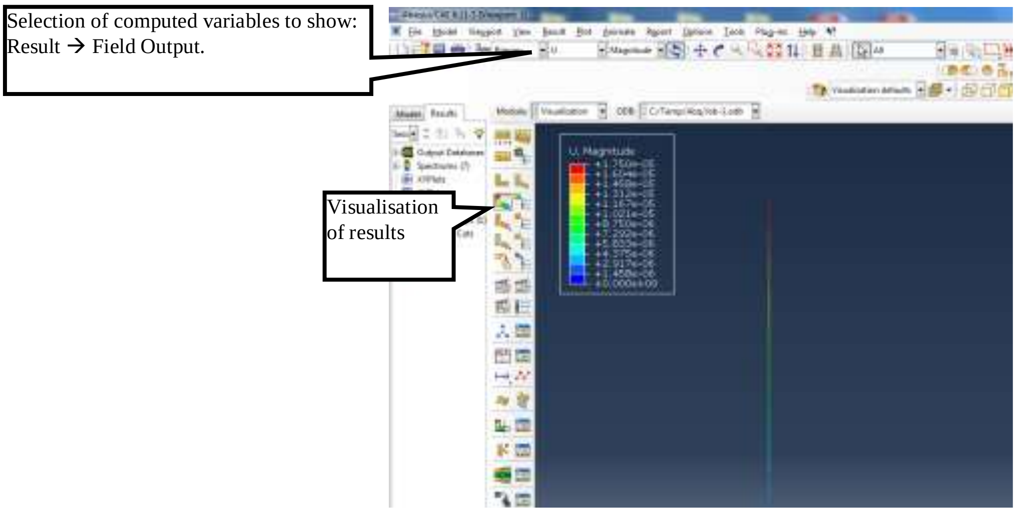
Create a new run called « tensile ». Generate the input file by the means of a button *Write input* ; Then, submit the run through the button *Submit*.

You can read the status of the run, if the status is "complete", everything is OK and you can see the computed results (in the next module), else go the *Monitor* menu, to identify the *errors* in the model.

Finally, go to the results by clicking on *Results*.

Visualization Module

Display the different results and compare them with those expected (see Theoretical study of tensile test)



Conclusion…

Store of the results.

Complementary analysis

1. Return to the Mesh module, and mesh the part with a single element. Run the calculation, view the results. Conclusion

2. Return to the Mesh module and mesh with 100 elements. Conclusion.

9.7.4 Shell Model

It is proposed to carry out a tensile test this time with shell "shell" elements.

Create a new file called "Tensile_shell".

Part Module
Select *3D*, *Deformable*, *Shell*, *Planar*.
Draw a rectangular shape corresponding to the front side of the part (140 x 20 mm).

Property Module
Define the material with the same option as previously.

Menu *Create Section*, Name : *Eprouvette*, Option *Shell*, *Homogeneous*. Then, define the thickness of the shell : *1 mm*, and attribute these properties to the part.
Menu *Assign Section*, select the specimen.

Assembly Module
Select the *Independant* option (the mesh will be created through the Mesh menu).

Step Module
Same steps as for the previous analysis.

Load Module
Define the loading and boundary conditions.

Mesh Module
Leave the default size for seeds. Mesh.

Job Module
Same steps as for the previous analysis.

Visualization Module
Comment the results. Compared with those previously obtained.

Complementary studies
1. Decrease the element size of the mesh. Restart the calculation. Conclusion.

2. Set up a very regular square mesh (3 mm wide). Use the mesh *structured* option in the *Assign Mesh Control* menu. Restart the calculation. Conclude.

9.7.5 Solid Model

Module part
3D, Solid, Deformable, Option extrusion. Draw the cross section of the specimen and extrude it from the value of the height of the specimen.

The rest of the steps are almost similar than the previous ones following for the Shell. Attention to specific options (*material option, definition of boundary conditions*…)

Analysis of the main results.

9.8 Analysis of the embedded conditions

View the stress values at the load application zone. Does that seem normal to you?
View the detail of the stress state located on the embedding zone. What do you think of that? What do you propose? Implement it.

9.9 Compressive simulation

Take the model and adapt it to make a compression test with a force of 200N.

Start the calculation and look at the results.

What do you think of that? Does this seem realistic?

10 Various case studies

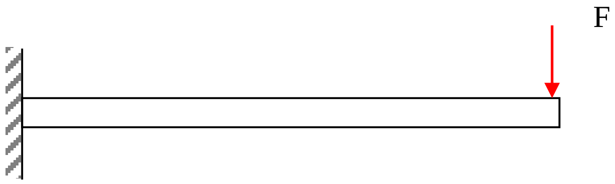
10.1 Beam study

We want to study a very simple case, the performances of a beam element “B31” and “B32”. The B31 and 32 elements are planar beam elements: 2 nodes at the ends and 3dof per node (Ux, Uy and Rz). We can therefore apply forces to them according to X, Y and a moment according to Z.

The beam to be studied is shown below. It is made of steel material (210GPa, ν=0.28), rectangular in cross-section and subjected to a concentrated force F.

In the case of a bending beam, the following relationship can be written:

$$y''(x) = \frac{Mfz(x)}{EI}$$



Geometry of the specimen and loadings :
b=0.1m ;
L=4m ;
F=2000N ;
h= 0.15m

10.1.1 Study of a thin beam subjected to concentrated load

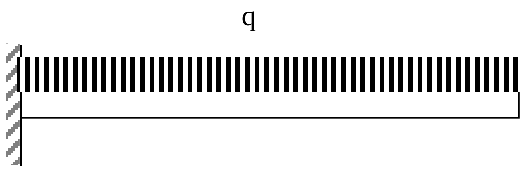
Q1. From the relationship between the value of the bending moment and the second derivative of the vertical position, give the theoretical value of the displacement. What should be the order of the polynomial that would give an exact interpolation of the displacement.

Q2. Set up a finite element model using only one B21 element. Read the value of the calculated displacement.

Q3. Set up a finite element model using eight B21 elements. Then eighty B21 elements. Compare the values obtained and the accuracy against the calculated theoretical value.

Q4. Follow the same approach with B22 elements with 1, 8 and 80 elements. Conclude.

10.1.2 Study of a thin beam subjected to linear loading



Geometry and loads:
b=0.1m ;
L=4m ; q=500N/m ; h= 0.15m

10.1.2.1 Simulation with only one Beam element (B21 & B22)

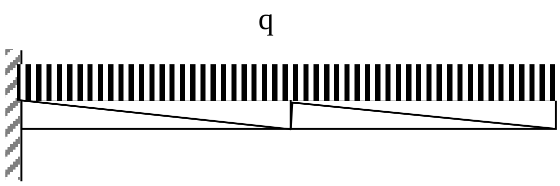
Q1. From the relationship between the value of the bending moment and the second derivative of the vertical position, give the theoretical value of the displacement. What should be the order of the polynomial that would give an exact interpolation of the displacement.

Q2. Store the different obtained results for 8 and 16 elements. Conclude.

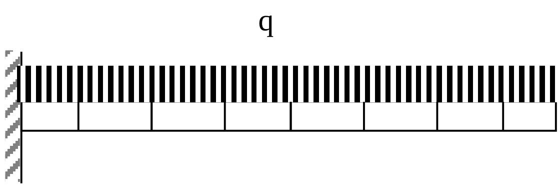
10.1.2.2 Set up of a 2D model

The objective is to build the same problem using 2D elements. Several meshes are proposed. For each of them, analyze and compare the results and then draw conclusions on the validity of the used meshes.

Case 1. Mesh with 4 triangular elements: (CPS6M: 6-nodes and 2dof)

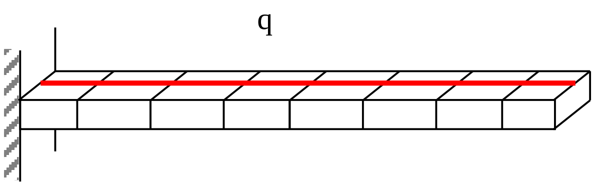


Case 2. Mesh with 8 triangular elements: CPS4 and CPS8 (4 or 8 nodes, 2dof)



10.1.2.3 Set up of 3D model

Set up a model with a single layer of elements in the thickness and 6 elements in the length. The elements to be used are: C3D8 (8 nodes with 3dof). Compared with the previous results.



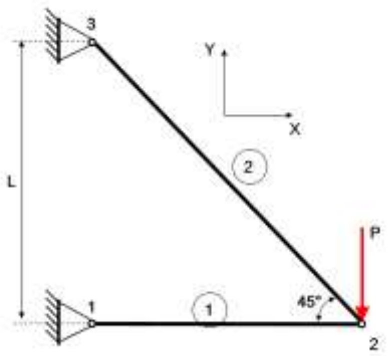
10.1.3 Creating a flat articulated support

The creation of an articulated structure can be done in different ways. It is possible to create mechanical connections between elements (ball joint, pivot, point...). To do this, options in the "interaction" menu of Abaqus allows you to declare them.

Another approach is based on identifying the degrees of freedom of the selected elements. For example, if you want to make a ball joint, you can use bar elements for which there are hinges at each end. We propose on a simple case to present these two approaches:

- Using the properties of the elements
- Creating a link between elements

10.1.3.1 Case study



The material used is a polycarbonate plastic with the following characteristics:

E = 2400 MPa, ν=0.3

Geometrical dimension : L = 200 mm and P=10N

The cross section has a circular shape with a diameter of 5 mm.

At each end, there is a pivot connection.

10.1.3.2 By the means of element property

By analyzing the load modes of this structure, we can see that part 2 works in tension and part 1 in compression. In the course, we identified elements that had hinges at each end and were working in tension/compression (the truss element).

In order to use these properties in the creation of a model on Abaqus, the following specifications are proposed:

1. In the *Part* menu: create a unique part regrouping the two frames 1 and 2.
2. In the P*roperty* menu: define the property of the *truss*.
3. The next step of the model definition remains the same as for other analysis.
4. Define the boundary conditions to the nodes number 1 and 3.
5. In the *Mesh* menu: define on both the zone of the structure (zones 1 and 2), only one truss element for each. This operation will generate a ball joint between each of the parts.
6. Run the calculation and conclude on the relevance of the results between the model and the imposed solicitations.

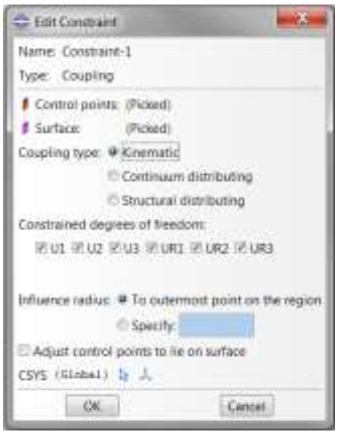
10.1.3.3 Definition of mechanical joints between elements

We used beam elements to model both tension/compression, bending and shear. All degrees of freedom are locked at each end of the element. In order to impose a pivot joint, it is necessary to make a connection with 2 parts and to declare a pivot joint in point 2.

For this purpose, it is proposed to:

1. In the *Part* menu: create 2 separate parts
2. In the P*roperty* menu: define *beam* property.
3. In the *Assembly* menu: make the assembly of the 2 parts.

4. In the *Interaction* menu: define the kinematic constraints (*create constraint*). Select the joint type (*Coupling*), then, select the 2 points corresponding to the intersection between the 2 parts. (position of the joint at point number 2).



We want to create a kinematic type coupling, by requiring that the Ui translations are identical between the 2 parts.
Select the appropriate options in the dialog box as shown on the left.

5. Define the boundary conditions in points 1 and 3.
6. In the *Mesh* menu: define the characteristic of the beam element (parts 1 and 2).
7. Run the simulation and conclude on the obtained results.

10.1.3.4 Additional computation and analysis

We propose, in order to train you (for the exam), to carry out the theoretical study of this assembly (2 bars), one of which is inclined.

According to the literature, we can explain the stiffness matrix of the inclined frame oriented by an angle θ:

$$[Ke] = \frac{ES}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}$$

In our case, the angle is equal to 135° (the horizontal reference corresponds to the x axis oriented on the right), we finally obtain:

$$[K_2] = \frac{ES_2}{2L_2}\begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix}$$

From the previous data:

1. Express the stiffness matrices of each bar (horizontal and inclined) + lengths.
2. Assemble the different stiffness matrix to form the global stiffness matrix of the structure.
3. Define the adapted boundary conditions
4. Solve the equations and compare with the results obtained during the practical analysis (through Abaqus).

Some correction elements:
For each bar the lengths are respectively: L1=200mm ; L2= 200/(*sin* 45°) = 282.8mm
Assuming that $k2 = ES_2/2L_2$ and $k_1=ES_1/L_1$, we can define the following global stiffness matrix:

$$\mathbf{K}\boldsymbol{ass}= \begin{vmatrix} \boldsymbol{k1} & -\boldsymbol{k1} & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ -\boldsymbol{k1} & \boldsymbol{k1}+\boldsymbol{k2} & -\boldsymbol{k2} & -\boldsymbol{k2} & \boldsymbol{k2} \\ \mathbf{0} & -\boldsymbol{k2} & \boldsymbol{k2} & \boldsymbol{k2} & -\boldsymbol{k2} \\ \mathbf{0} & \boldsymbol{k2} & -\boldsymbol{k2} & -\boldsymbol{k2} & \boldsymbol{k2} \end{vmatrix}$$

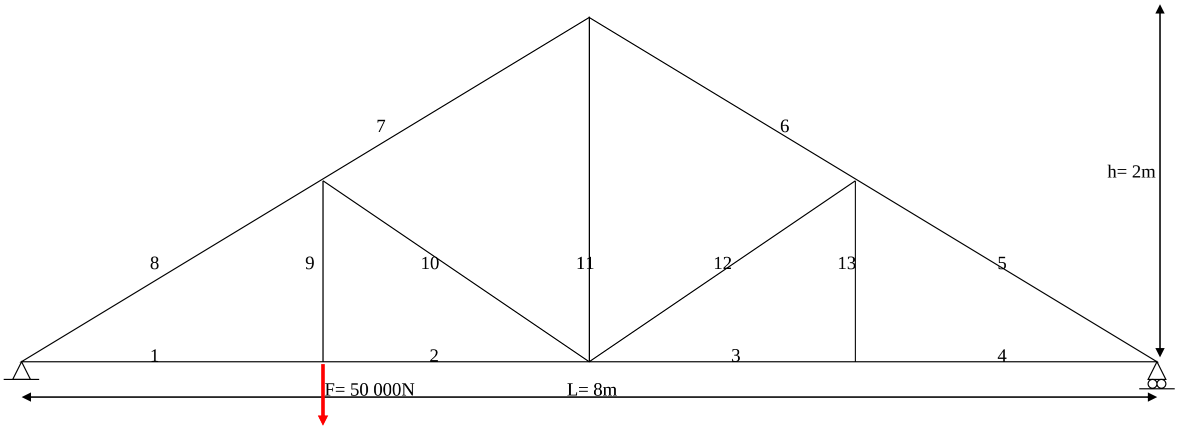
By considering the boundary condition, we can obtain:
Limit cond.. : u1=u3=v3=0 ;
F2y = -P ;

$$\begin{cases} (k1+k2)u2 - k2v2 = 0 \\ -k2u2 + k2v2 = -\mathrm{P} \end{cases} \Rightarrow \mathrm{u2} = \frac{-\mathrm{P}}{k1} \text{ and } \mathrm{v2}= \frac{-p}{k2}\left(1+\frac{k2}{k1}\right)$$

You can directly compare the present values to the values computed by the means of Abaqus.

10.1.4 Lattice / Gantry structure

The objective of this study is to study the deformation and stresses in the bars or beams of a lattice or gantry structure, subject to planar loading



We consider 2 different cases with the same kind of loads :

- Structure A. lattice structure composed of *articulated* bars
- Structure B. gantry structure composed of welded bars.

Les bars 1 to 8 are made of tubes with an external diameter of 60.3mm and a thickness of 2.9mm.
The bars 9 to 13 are made of tubes with an external diameter of 21.3mm and a thickness of 2.3mm.
The material for all tubes is steel (E=210GPa, ν=0.3)

10.1.4.1 Work to be done

On structure A:

What type of element can be chosen to mesh the structure?
Compare the reactions to the supports calculated with Abaqus and analytically
Read the vertical displacement of node n2.
Read the axial forces in each of the bars. What do you think of that?

On structure B :

What type of element can be chosen to mesh the structure?
How can we check that the bars are welded together?
Read the vertical displacement of node n2 and the axial forces in each of the bars. Compared with the values found above?
Remove the least stressed beam. Conclusion.
What happens if the number of elements is increased (e.g. 5 elements per bar).