Seminar on MEC423 - Finite Element for Deformable Bodies

By Henri Champliaud & Lê Văn Ngàn
Professors, École de technologie supérieure, Montréal, Canada

INTRODUCTION

* Growing need of Finite Element Specialists: Due to competition, industry needs more and more engineers with solid background of theoretical and practical Finite Element Method so as to apply Finite Elements with judgement for avoiding misinterpretation in analyzing structures, assemblies, manufacturing processes, etc. That is why Finite Element Method need taught during undergraduate engineering programs.

* Dilemma of teaching Finite Element Method (FEM): Teaching classical FEM involves a lot of time spent for heavy mathematics so that average students could not retain much about FEM after attending the course. On the other hand, if one just teaches how to use a FEM software (most of students just want that), it would be dangerous for industry because they would lack knowledge about several essential theoretical aspects of FEM and not be able to realize mistakes and to identify which results to be used.

* A proposed compromise for undergraduate engineering course about FEM:

1. Title of the course : "Finite Element Method for Deformable Bodies". Any other title could be acceptable, but must show that it is limited to solid bodies, not for fluids.

2. Prerequisite knowledge : The students must have acquired following subjects: Integral calculus, matrix algebra and basic strength of materials. Although basic notions about heat transfer by conduction are needed but the complete course on heat transfer is not a prerequisite for FEM because it just takes about one hour in the FEM course to introduce essential relationships of common sense about heat conduction and convection.

3. Alternation between theories and practices : In each typical week, there are about 3 hours teaching a theoretical subject by FEM including 2 to 3 examples to be solved by hand for better understanding the theory, followed (one or few days later) by 2 to 3 hours of practicing a FEM software for solving simplified but practical problems related to the theoretical subjects, the practical problems being prepared with enough guide details so that an average student could do it by himself and complete the answers within 2 hours. This scenario is repeated for about 13 weeks on about 6 to 7 selected main subjects.

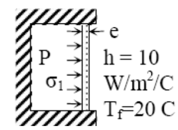
Subjects and teaching approach for undergraduate engineering course on FEM

Chapter 1 : Revision on integrals and matrix algebra (subjects not shown)

Chapter 2. FEM for heat conduction (6 h class + 4 h hands on FEM software)

[1 hour]- Temperature gradient $\vec{\varepsilon} = \{\partial T/\partial x; \partial T/\partial y; \dots\}$; heat conduction coefficient k, heat flux by conduction $\vec{\sigma}=-k\vec{\varepsilon}$, heat flux by convection $\vec{\sigma}\cdot\vec{n}=h(T-T_f)$, steady state equilibrium equation; followed by an example like 2.1.

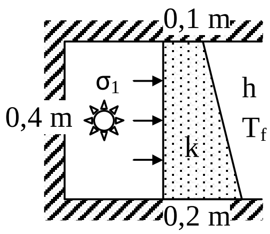
Exemple 2.1 : Une puissance thermique P = 1000 W doit traverser la surface S = 4 m^2 d'un mur d'épaisseur e = 0.025 m. Le mur est refroidi par une convection telle que montrée. Calculer les tempéatures sur les deux faces du mur s'il est en (1) Acier innoxydable 304; (2) Plexiglass.



No problem for students to understand these concepts before the heat transfer course.

[2 hours]- Introduction to the principle of stationary functional of heat conduction,

$$\Pi = \int_V \tfrac{1}{2} k_c\, \varepsilon^2\, dV + \int_{S_h} \left(\tfrac{1}{2} h T^2\right) dS - \int_{S_h} \left(h T_f T\right) dS - \int_V \left(Q T\right) dV - \int_{S_1} \left(\sigma_1 T\right) dS \quad \text{must be stationary.}$$

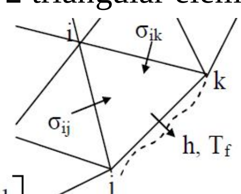


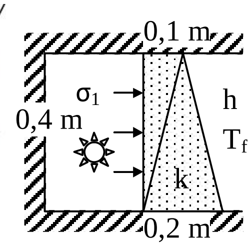
Example: A plane piece as shown with material k = 5 W/(m·°C) is subjected to a heat flux σ_1 = 3000 W/m^2 on the left boundary and air convection h = 10 W/(m^2·°C), T_f = 20 °C on right boundary. By using the minimum functional Π, determine the max and min temperatures in the body using the model $T = c_1 + c_2\cdot x + c_3\cdot y$.

This example could be done by hand within 30 minutes. The advantage of heat conduction is that there are several simple but interesting examples like this for student to practice and better understand the stationary principle of boundary condition problems (in comparison with structural displacement models such as $U_X = c_1 + c_2\cdot x + c_3\cdot y$ and $U_Y = c_4 + c_5\cdot x + c_6\cdot y$, which would take more than 2 hours to solve the problem by hand).

[3 hours]- Developing the stationary principle Π for plane and axisymmetrical triangular elements gives explicit 3x3 matrix equations per element, allowing students to practice hand the assembly principle of FEM using models of 2 to 4 triangular elements. Example: Solve the previous example using a model of 2 triangular elements.

Equilibrium equation for a plane triangular element i-j-k :





$$\boxed{[K]\{D\} + [H]\{D\} = \{f_h\} + \{f_v\} + \{f_\sigma\}} \quad \dots\dots\dots (2.18)$$

$$\mathbf{[K]} = \frac{k_c\, e}{4\,|A|}\begin{bmatrix} \vec{V}_1\cdot\vec{V}_1 & \vec{V}_1\cdot\vec{V}_2 & \vec{V}_1\cdot\vec{V}_3 \\ \vec{V}_2\cdot\vec{V}_1 & \vec{V}_2\cdot\vec{V}_2 & \vec{V}_2\cdot\vec{V}_3 \\ \vec{V}_3\cdot\vec{V}_1 & \vec{V}_3\cdot\vec{V}_2 & \vec{V}_3\cdot\vec{V}_3 \end{bmatrix} \text{ pour } \{D\}=\begin{bmatrix} T_i \\ T_j \\ T_k \end{bmatrix}; \ \mathbf{[H_{jk}]} = \frac{h\, e\, L_{jk}}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}$$

Pour $\{D\} = \begin{bmatrix} T_j \\ T_k \end{bmatrix}$

$$\{f_V\} = \frac{Q\, e\, |A|}{3}\begin{bmatrix} 1 \\ 1 \\ 1 \end{bmatrix}, \quad \left\{f_{h(j\text{-}k)}\right\} = \frac{h\, T_f\, e\, L_{jk}}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix}, \quad \left\{f_{\sigma(i\text{-}j)}\right\} = \frac{\sigma_{ij}\, e\, L_{ij}}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} \quad \dots\dots\dots\dots (2.19)$$

Aux nœuds où la convection est connue — Aux nœuds où le flux est connu

$\boxed{\bar{V}_1 = \text{j vers k}}$, c.à.d. $[(x_k-x_j); (y_k-y_j); 0]$; $\boxed{\bar{V}_2 = \text{k vers i}}$; $\boxed{\bar{V}_3 = \text{i vers j}}$; e = épaisseur

Chapter 3. FEM for 2D truss structures (6 h class + 4 h hands on FEM software)

[3 hours]- The first half of this chapter is to introduce the stationary functional principle for structures,

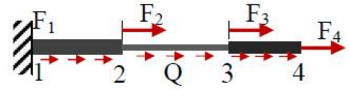
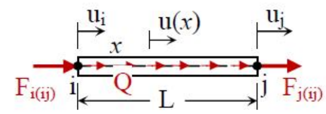


Fig. 3.5.a - Structure uniaxiale

$$\Pi = \int_V \left(\tfrac{1}{2}[\varepsilon]^t[C][\varepsilon] - [\varepsilon]^t[C][\varepsilon_T] - [u]^t[Q]\right)dV - \int_S [u]^t[p]dS$$



and to apply it to **1D spar structures** as shown.

- Équations d'équilibre d'un élément « tige 1D » :

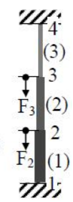
$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = AE\alpha\Delta T\begin{Bmatrix} -1 \\ 1 \end{Bmatrix} + \frac{ALQ}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} + \begin{Bmatrix} F_{i\,(ij)} \\ F_{j\,(ij)} \end{Bmatrix} \quad (3.13)$$

L : longueur ; A : aire de section
Q : force volumique (N/m³)
E : module d'élasticité
A: coeff. de dilatation thermique

Matrice de rigidité $[K]\{u_{ij}\} = \{f_T\} + \{f_Q\} + \{f_{interne}\}$ inconnues au départ

This subject is classic, but mechanical and civil students should practice examples like this to realize important things:

Exemple 3.1 : Modèle de trois éléments tige 1D : $L_1 = L_2 = L_3 = 500$ mm, un seul matériau dont E = 10000 MPa, α = 20E-6 /°C et Q = -24000 N/m³, sections $A_1 = 800$ mm², $A_2 = 400$ mm² et $A_3 = 100$ mm². Les deux bouts sont bloqués.
(a) Construire les équations des éléments et les équations d'assemblage complètes.
(b) Déterminer les déplacements, les forces de réactions, les forces internes et les contraintes aux noeuds dues au poids, ΔT = 25°C, $F_2 = 3000$ N↓ et $F_3 = 4000$ N↓;
(c) Vérifier l'équilibre des forces : de réaction, sur les éléments 1, 3, et au nœud 2.



*Assembly equations and support conditions are essential for calculating displacements and reaction forces; *Individual equations are reused in post treatment for calculating internal forces and stresses; * Verification of results is a must before accepting them.

[3 hours]- Equations of 2D spar are obtained by transformation equations of 1D spar.

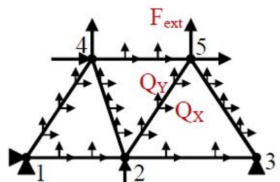
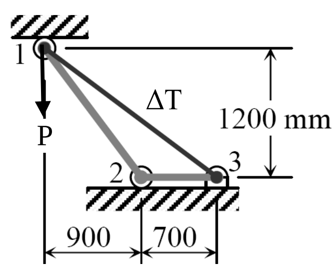


Fig.3.8 - Structure de treillis

$$\frac{A\cdot E}{L}\begin{bmatrix} c^2 & s\cdot c & -c^2 & -s\cdot c \\ s\cdot c & s^2 & -s\cdot c & -s^2 \\ -c^2 & -s\cdot c & c^2 & s\cdot c \\ -s\cdot c & -s^2 & s\cdot c & s^2 \end{bmatrix}\begin{bmatrix} U_{xi} \\ U_{yi} \\ U_{xj} \\ U_{yj} \end{bmatrix} = A\cdot E\cdot\alpha\cdot\Delta T\begin{bmatrix} -c \\ -s \\ c \\ s \end{bmatrix} + \frac{A\cdot L}{2}\begin{bmatrix} Q_x \\ Q_y \\ Q_x \\ Q_y \end{bmatrix} + \begin{bmatrix} F^{i\text{-}j}_{xi} \\ F^{i\text{-}j}_{yi} \\ F^{i\text{-}j}_{xj} \\ F^{i\text{-}j}_{yj} \end{bmatrix}$$

This subject is classic but need be emphasized on the followings: 2 degrees of freedom per node (U_{Xi} and U_{Yi}), 2 nodes per element, 4x4 matrix equilibrium equations per element, assembly equations, support conditions, reaction forces, internal forces (F^{ij}_{Xi}, etc.), how to calculate axial force F_{ij} at node i and j.



Example 3.2: Given P = 800 N, ΔT = 25°C, $A_{12} = A_{23} = 100$ mm₂, $A_{13} = 50$ mm², E = 200000 MPa, α = 12.5E-6 /°C. Using a model with three 2D spar elements, calculate reaction forces and axial stresses in each bar.

Chapter 4. FEM for beam structures (6 h class + 4 h hands on FEM software)

[3 hours]- Study 1D beam models.

The example 4.2 takes 30 minutes of hand work.

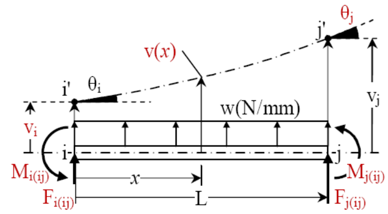


Figure 4.3 - Élément poutre 1D

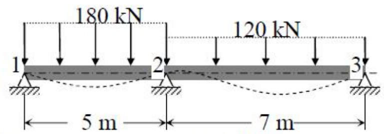
$$[k]\{D_{ij}\} = \{f_{w(ij)}\} + \{f_{int.ij)}\}$$

$$\begin{bmatrix} K_2 & K_3 & -K_2 & K_3 \\ K_3 & 2K_4 & -K_3 & K_4 \\ -K_2 & -K_3 & K_2 & -K_3 \\ K_3 & K_4 & -K_3 & 2K_4 \end{bmatrix} \begin{Bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{Bmatrix} = \begin{Bmatrix} Lw/2 \\ L^2w/12 \\ Lw/2 \\ -L^2w/12 \end{Bmatrix} + \begin{Bmatrix} f_i^{i\text{-}j} \\ M_i^{i\text{-}j} \\ f_j^{i\text{-}j} \\ M_j^{i\text{-}j} \end{Bmatrix} \ldots\ldots$$

$\int_A y^2 dA$; $K_2 = \frac{12 \cdot E \cdot I}{L^3}$; $K_3 = \frac{6 \cdot E \cdot I}{L^2}$; $K_4 = \frac{2 \cdot E \cdot I}{L}$

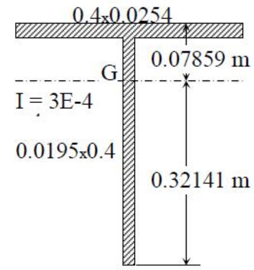
It is suggested to give partial results and ask for completing the remaining, chosen so that students learn important principles and could find answer with about 30 minutes.

Exemple 4.2 : La poutre 1-2-3 est en acier dont E = 200E6 kN/m^2, supportée et chargée tel que montré. les dimensions, le centroïde G et le moment d'inertie I de section sont tels que donnés.



En modélisant la poutre en deux éléments « poutre 1D »,

Compléter les espaces vides ci-dessous, construire les équations d'assemblage, calculer les contraintes max et min.

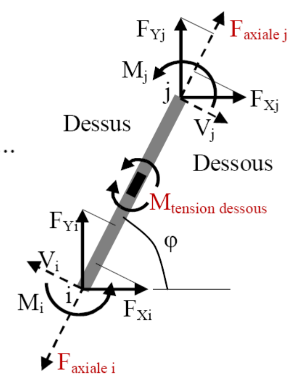


$$\begin{bmatrix} 2099.1 & 7346.9 & -2099.1 & 7346.9 \\ 7346.9 & 34286 & -7346.9 & 17143 \\ -2099.1 & -7346.9 & 2099.1 & -7346.9 \\ 7346.9 & 17143 & -7346.9 & 34286 \end{bmatrix} \begin{Bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} -60 \\ -70 \\ -60 \\ 70 \end{Bmatrix} + \begin{Bmatrix} \cdots \\ \cdots \\ \cdots \\ \cdots \end{Bmatrix} \quad (2\text{-}3)$$

$$\begin{bmatrix} 5760 & \cdots & -5760 & \cdots \\ \cdots & \cdots & \cdots & 24000 \\ -5760 & \cdots & 5760 & \cdots \\ \cdots & 24000 & \cdots & \cdots \end{bmatrix} \begin{Bmatrix} \cdots \\ \cdots \\ \cdots \\ \cdots \end{Bmatrix} = \begin{Bmatrix} \cdots \\ \cdots \\ \cdots \\ \cdots \end{Bmatrix} + \begin{Bmatrix} \cdots \\ \cdots \\ \cdots \\ \cdots \end{Bmatrix} \quad (1\text{-}2)$$

[3 hours]- 2D beam structures

$$\begin{bmatrix} K_5 & K_6 & -K_8 & -K_5 & -K_6 & -K_8 \\ K_6 & K_7 & K_9 & -K_6 & -K_7 & K_9 \\ -K_8 & K_9 & 2K_4 & K_8 & -K_9 & K_4 \\ -K_5 & -K_6 & K_8 & K_5 & K_6 & K_8 \\ -K_6 & -K_7 & -K_9 & K_6 & K_7 & -K_9 \\ -K_8 & K_9 & K_4 & K_8 & -K_9 & 2K_4 \end{bmatrix} \begin{Bmatrix} U_{Xi} \\ U_{Yi} \\ \theta_i \\ U_{Xj} \\ U_{Yj} \\ \theta_j \end{Bmatrix} = AE\alpha\Delta T \begin{Bmatrix} -c \\ -s \\ 0 \\ c \\ s \\ 0 \end{Bmatrix} + \begin{Bmatrix} Lw_X/2 \\ Lw_Y/2 \\ L^2(w_Y c - w_X s)/12 \\ Lw_X/2 \\ Lw_Y/2 \\ -L^2(w_Y c - w_X s)/12 \end{Bmatrix} + \begin{Bmatrix} F_{Xi}^{ij} \\ F_{Yi}^{ij} \\ M_i \\ F_{Xj}^{ij} \\ F_{Yj}^{ij} \\ M_j \end{Bmatrix} \ldots$$

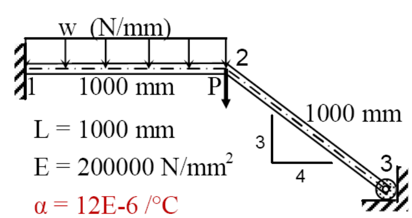


Symboliquement: $[K] \quad \{D_{i\text{-}j}\} = \{F_v = F_T + F_w\} \quad + \{F_{int}\}$

Où $K_1 = A \cdot E/L$; $K_2 = 12 \cdot E \cdot I/L^3$; $K_3 = 6 \cdot E \cdot I/L^2$; $K_4 = 2 \cdot E \cdot I/L$;

$K_5 = K_1 * c^2 + K_2 * s^2$; $K_6 = (K_1 - K_2) * s * c$; $K_7 = K_1 * s^2 + K_2 * c^2$; $K_8 = K_3 * s$; $K_9 = K_3 * c$

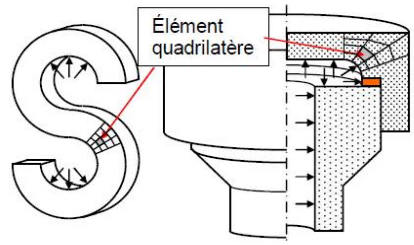
This subject is classic but examples are too long if doing from A to Z. It is suggested to give partial results and just ask for completing the remaining, chosen such that students could do an example within 30 minutes and learn more important things on how to calculate axial force (F_{axial}), internal bending moment (M) and combined stresses at top and bottom $(\frac{F_{axial}}{A} \pm \frac{M \cdot c}{I})$.



w = 18 N/mm, P = 10000 N, ΔT = 25°C. Compléter les espaces vides, construire les équations d'assemblage réduites, calculer les contraintes combinées max et min.

Chapter 5. Isoparametric formulation (6 h class + 4 h hands on FEM software)

Since related mathematics become heavy and constitutive relations are long to manipulate by hand (8x8 matrices for just one element, etc.), we must skip some routines already practiced in previous chapters (such as assembly, solving for displacements …) and spend time for accuracy convergence and interpretation of results.



[3 hours]- Quadrilateral element formulations**:** Theoretical concepts about interpolation using normalized coordinates r, s, jacobian matrix, strain-displacement and stress-strain relationships need be learned and practiced to understand how stresses at a point are computed by knowing displacements at nodes I,J,K,L such as example 5.5.

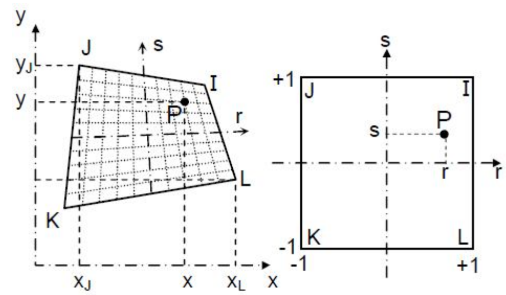


Figure 5.1 Coordonnées normalisées « r, s » dans un quadrilatère IJKL

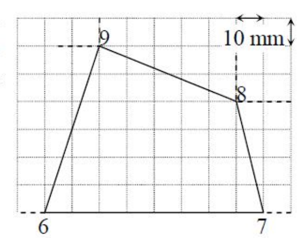
[3 hours]- Accuracy, convergence, singularity, linerized stresses, interpretation of solid model results:

Many engineers do not verify results of solid finite element models and often incorrectly use them because most of FEM books do not insist enough on them.

In this example, stresses converge at A but are singular at D (→∞) due to force at one point.

Exemple 5.5 : Une pièce plane modélisée en éléments IJKL d'état plan de contrainte donne les résultats résumés dans le tableau ci-dessous. Les propriétés du matériau sont E = 2002 MPa, ν = 0.3 et α = 120∗10^{-6} /ºC.

Noeud	x(mm)	y(mm)	ΔT(C)	u_x (mm)	u_y (mm)
6	100	0	20	0.075	0
7	180	0	18	0.100	0
8	170	40	12	0.050	0.050
9	120	60	14	0.10	0.025

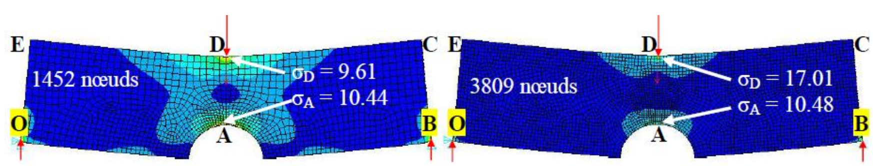


(a) Dessiner la déformée de l'élément 8-9-6-7 en amplifiant les déplacements de 100 fois.

(b) Calculer les jacobiennes, les déformations et les contraintes au centre de cet élément.

(c) Vérifier chacun des termes des matrices jacobiennes à chaque point.

Rép. (b) : [σ_x ; σ_y ; τ_{xy}] Nœud 8 = [-6.7767 ; -1.8573 ; -0.4278] MPa ;
Nœud 9 = [-6.2769 ; -4.8048 ; 0.9625] MPa ; Point C = [-5.5162 ; -4.0579 ; 0.05833] MPa

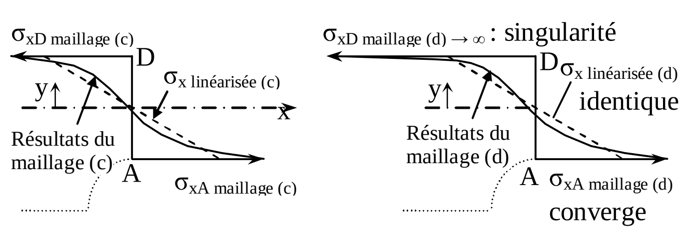


(c) Maillage gaspillé loin de la région A (d) Maillage gaspillé loin de la région A

Nominal stresses by linearization must be done and used for most of design criteria:

$$F = \int_{y_A}^{y_D} \sigma \cdot dy; \quad M = -\int_{y_A}^{y_D} \sigma \cdot y \cdot dy$$

$\sigma_{x\ linarized} = \frac{F}{A} - \frac{M \cdot y}{I}$ which are practically unaffected by mesh density.



Chapter 6. Structural boundary equations (6 h class + 4 h hands on FEM software)

General principle of boundary equations with FEM:

For n assembly equations, there are 2n values to be determined.

$$[K]_{nxn}\{u\}_{nx1} = \{F_V\}_{nx1} + \{F_S\}_{nx1} + \{F_{ext}\}_{nx1} \quad \text{……..(6.1)}$$

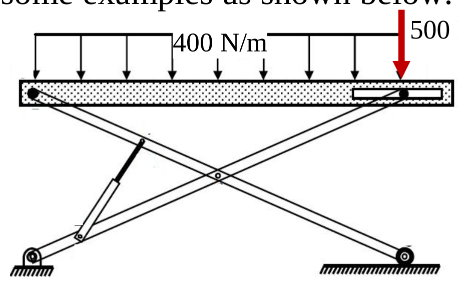
n equations with 2n values to be determined

$[K]$: known; $\{u\}$: n values to be determined; $\{F_V\}$, $\{F_S\}$: known; $\{F_{ext}\}$: n values to be determined

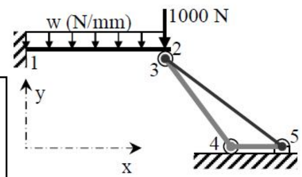
Hence, there must be **n boundary equations** to be identified in order to solve (6.1).

Some theoretical concepts need to be learned for doing some examples as shown below.

Example 6.1: (a) Describe a model (number of nodes and element connectivity) with minimum number of 2D beam elements for studying the scissor elevator as shown; (b) identify all boundary equations and underline those which are not essential.



Exemple 6.2 : Le modèle d'un élément « poutre 1D » et de trois éléments « tige 2D » a les équations d'assemblage partiellement réduites ci-dessous avec unités N et mm.



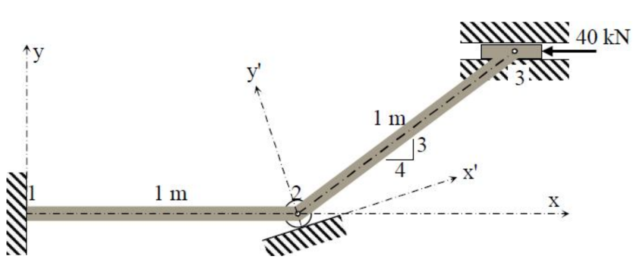
$$\begin{bmatrix} 240 & -12e4 & 0 & 0 & 0 \\ -12e4 & 8e7 & 0 & 0 & 0 \\ 0 & 0 & 3520 & -4320 & -2880 \\ 0 & 0 & -4320 & 5480 & 3840 \\ 0 & 0 & -2880 & 3840 & 12880 \end{bmatrix} \begin{Bmatrix} u_{y2} \\ \theta_2 \\ u_{x3} \\ u_{y3} \\ u_{x4} \end{Bmatrix} = \begin{Bmatrix} -6e2 \\ 1e5 \\ 0 \\ 0 \\ 0 \end{Bmatrix} + \begin{Bmatrix} \\ \\ \\ \\ \\ \end{Bmatrix}$$

Compléter les variables du dernier vecteur, construire les équations réduites en fonction du contact aux nœuds 2 et 3 et de la force de 1000 N appliquée sur ces nœuds, résoudre pour les déplacements / rotations et calculer les forces nettes sur chaque nœud 2 et 3.

Example 6.3:

L'assemblage ci-dessous est modélisé en deux éléments poutre 2D : 1-2 et 2-3. Le bout 1 est encastré, le manchon soudé au bout 3 est guidé sans rotation et sans frottement dans une rainure horizontale et le coude 2 est supporté sur un plan inclinable. L'ensemble est chauffé de 100 ºC et une force de 40 kN est appliquée au bout 3 vers la gauche. Les équations d'assemblage, partiellement réduites après l'élimination des déplacements rotations u_{x1}, u_{y1}, θ_1, u_{y3} et θ_3, sont données ci-dessous, les unités sont kN, mm.

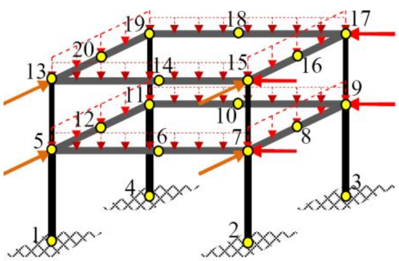
Si le plan de blocage au coude 2 est parallèle à 2-3, construire les équations réduites, déterminer les résultats des déplacements et la force de réaction au coude 2.



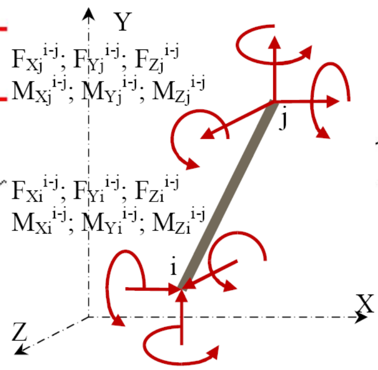
$$\begin{bmatrix} 332320 & 90240 & -3600 & -132320 \\ 90240 & 91680 & -1200 & -90240 \\ -3600 & -1200 & 8000 & 3600 \\ -132320 & -90240 & 3600 & 132320 \end{bmatrix} \begin{bmatrix} u_{x2} \\ u_{y2} \\ \theta_2 \\ u_{x3} \end{bmatrix} = \begin{bmatrix} 48 \\ -144 \\ 0 \\ 192 \end{bmatrix} + \begin{bmatrix} F_{x2} \\ F_{y2} \\ M_2 \\ F_{x3} \end{bmatrix}$$

Chapter 7. 3D FEM for structures (3 h class + 2 h hands on FEM software)

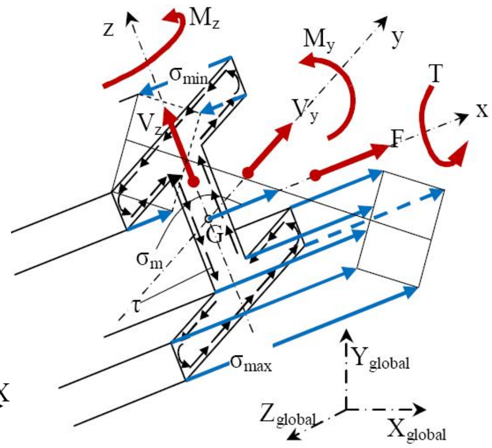
[1/2 hour]- 3D beam elements:



Example of 3D beam finite element model

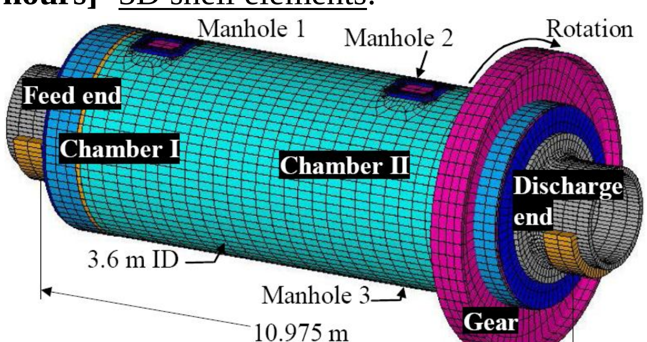


Internal forces and moments at nodes of a 3D beam element

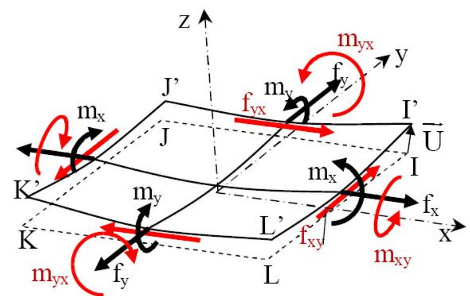


Local Gxyz system and stresses in a cross section of a 3D beam element

[2 hours]- 3D shell elements:



Example of shell-element model



Linear forces f_i, linear moments m_i and curvatures of plate and shell

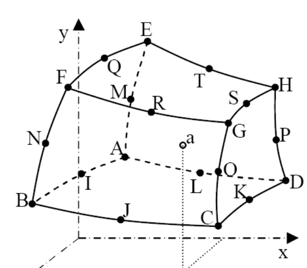
p_h = pression sur la face du haut (z=t/2) et p_b = pression sur la face du bas (z=-t/2)

Contraintes au dessus (z = t/2) : $\sigma_x = \frac{f_x}{t} - \frac{6 \cdot m_x}{t^2}$, $\sigma_y = \frac{f_y}{t} - \frac{6 \cdot m_y}{t^2}$, $\tau_{xy} = \frac{f_{xy}}{t} - \frac{6 \cdot m_{xy}}{t^2}$, $\sigma_z = -p_h$ . . . (a)

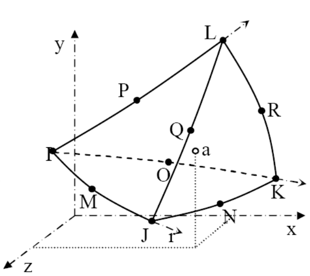
Contraintes au milieu (z = 0) : $\sigma_x = \frac{f_x}{t}$, $\sigma_y = \frac{f_y}{t}$, $\tau_{xy} = \frac{f_{xy}}{t}$, $\sigma_z = -0.5(p_h + p_b)$ . . . . (7.8.b)

Contraintes en dessous (z=-t/2) : $\sigma_x = \frac{f_x}{t} + \frac{6 \cdot m_x}{t^2}$, $\sigma_y = \frac{f_y}{t} + \frac{6 \cdot m_y}{t^2}$, $\tau_{xy} = \frac{f_{xy}}{t} + \frac{6 \cdot m_{xy}}{t^2}$, $\sigma_z = -p_b$ . . . . (c)

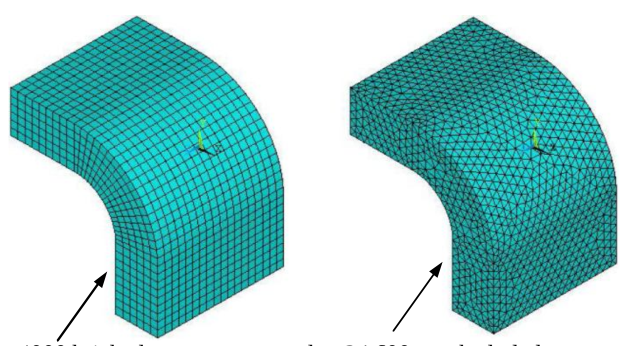
[1/2 hour]- 3D solid elements:



Brique 20 nœuds (BR20)



Tétraèdre 10 nœuds (TET10)

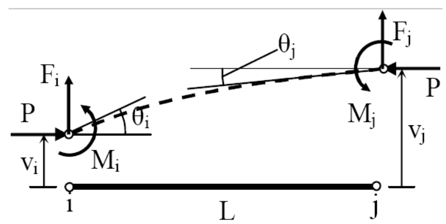

4000 brick elements compared to 24 600 tetrahedral elements

Chapter 8. Elastic buckling of beams (6 h class + 4 h hands on FEM software)

[3 hours]- 1D beam structures: Equations for 1 element including 2nd order stiffness matrix for large deflection:

$$\left[\begin{bmatrix} K_2 & K_3 & -K_2 & K_3 \\ K_3 & 2K_4 & -K_3 & K_4 \\ -K_2 & -K_3 & K_2 & -K_3 \\ K_3 & K_4 & -K_3 & 2K_4 \end{bmatrix} - P \begin{bmatrix} 1.2/L & 0.1 & -1.2/L & 0.1 \\ 0.1 & 4L/30 & -0.1 & -L/30 \\ -1.2/L & -0.1 & 1.2/L & -0.1 \\ 0.1 & -L/30 & -0.1 & 4L/30 \end{bmatrix}\right] \begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix} = \begin{bmatrix} F_i^{i\text{-}j} \\ M_i^{i\text{-}j} \\ F_j^{i\text{-}j} \\ M_j^{i\text{-}j} \end{bmatrix}$$

$\boxed{K_2 = 12EI/L^3}$; $\boxed{K_3 = 6EI/L^2}$; $\boxed{K_4 = 2EI/L}$



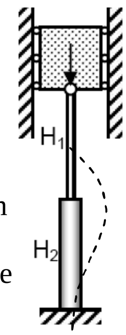
1D beam element with an axial force P in compression

Assembly equations during buckling = Eigen value & Eigen vector equations:

Buckling occurs when {U} indefinitely increases with no change of $\{F_{ext}\}$. Thus $\boxed{\left[[K_{t1}] - P_{cr}\cdot[K_{u2}]\right]\{\Delta U\} = \{0\}}$.

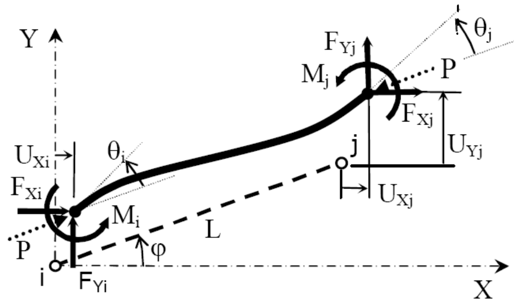
P_{cr} is the lowest Eigen value, and the buckling mode is the corresponding Eigen vector.

Example 8.2
Section 1 : H_1 = 8000 mm, D_{e1} = 100 mm, D_{i1} = 75 mm
Section 2 : H_2 = 8100 mm, D_{e2} = 175 mm, D_{i2} = 150 mm
Young's modulus : E = 200 000 N/mm^2
Using a model with two 1D-beam elements, calculate the critical load at cabin and draw the buckling mode.



[3 hours]- 2D beam structures:

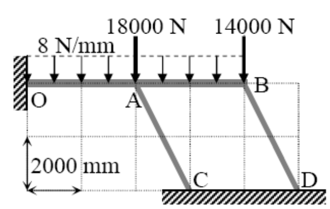
The 1st order element stiffness matrix is the same as in chapter 4; the unit 2nd order stiffness matrix of an element i-j is, with P = 1, is shown on right where $K_{s1} = 1.2/L$; $K_{s2} = 0.1$; $K_{s3} = L/30$; $c=\cos(\varphi)$, $s=\sin(\varphi)$; $K_{s4} = K_{s1}{*}s^2$; $K_{s5} = K_{s1}{*}s{*}c$; $K_{s6} = K_{s1}{*}c^2$; $K_{s7} = K_{s2}{*}s$; $K_{s8} = K_{s2}{*}c$;



$$[K_{2u}] = [\; K_{s4} \;\; -K_{s5} \;\; -K_{s7} \;\; -K_{s4} \;\; K_{s5} \;\; -K_{s7};\;\; -K_{s5} \;\; K_{s6} \;\; K_{s8} \;\; K_{s5} \;\; -K_{s6} \;\; K_{s8};\;\; -K_{s7} \;\; K_{s8} \;\; 4{*}K_{s3} \;\; K_{s7} \;\; -K_{s8} \;\; -K_{s3};\;\; -K_{s4} \;\; K_{s5} \;\; K_{s7} \;\; K_{s4} \;\; -K_{s5} \;\; K_{s7};\;\; K_{s5} \;\; -K_{s6} \;\; -K_{s8} \;\; -K_{s5} \;\; K_{s6} \;\; -K_{s8};\;\; -K_{s7} \;\; K_{s8} \;\; -K_{s3} \;\; K_{s7} \;\; -K_{s8} \;\; 4{*}K_{s3}];$$

General procedure of elastic buckling analysis:

1. Perform a static analysis: $[K_{t1}]\{U\} = \{F_w\} + \{F_{ext}\}$
2. Calculate axial forces P_i in elements; build the 2nd order assembly stiffness matrix:
   $[K_{t2}] = \Sigma_i\, P_{(i)}\cdot[K_{2u(i)}]$
3. Multiply all loads by an unknown factor f_{cr} to get buckling and solve $\left[[K_{t1}] - f_{cr}\cdot[K_{t2}]\right]\{\Delta U\} = \{0\}$ for f_{cr} and buckling mode.



The procedure 1, 2, 3 is very long. It is suggested to give enough partial results so that it could be done by hand within ¾ hour.

Example 8.3 - Young's modulus : E = 200 000 N/mm^2
A_1 = 1500 mm^2, I_1 = 800000 mm^4 for OAB ; A_2 = 1200 mm^2, I_2 = 500000 mm^4 for AC and BD. Using a model with one 2D-beam element in each segment, calculate the load multiplication factor for elastic buckling and draw the buckling mode.