Les éléments finis

MX-BA5

Mercredi 11 décembre 2024

Cours 11

Les déformations thermiques

*expansion

le couplage thermomécanique

*coupled temperature-displacement

Le fichier *.msg

Convergence et consistance

Pratique des éléments finis

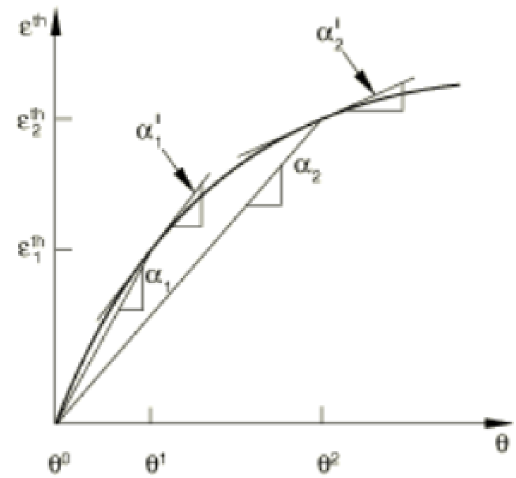
Sous l'effet de la température, les solides se dilatent ou se contractent.

CTE = coefficient of thermal expansion.

Dans le(s) bloc(s) matériaux:

*EXPANSION
Specify thermal or field expansion.

!!!!!! Abaqus suppose toujours $\varepsilon_{th} = 0$ initialement, i.e la géométrie de départ est non déformée à t = 0.



They generate thermal strains according to the formula

$$\varepsilon^{th} = \alpha(\theta, f_\beta)(\theta - \theta^0) - \alpha(\theta^I, f_\beta^I)(\theta^I - \theta^0),$$

where

$\alpha(\theta, f_\beta)$

is the thermal expansion coefficient;

θ

is the current temperature;

θ^I

is the initial temperature;

Pratique des éléments finis

Expansion thermique:

***EXPANSION**
Specify thermal or field expansion.

TYPE

Set TYPE=ISO (default) to define isotropic expansion. The only option that is available in an Abaqus/CFD analysis is TYPE=ISO.

Set TYPE=ORTHO to define orthotropic expansion.

Set TYPE=ANISO to define fully anisotropic expansion in an Abaqus/Standard analysis.

ZERO

If the thermal expansion is temperature- or field-variable-dependent, set this parameter equal to the value of θ^0. The default is ZERO=0.

***material, name=steel**
expansion, zero = 120.
******** CTE, temp**
12.e-6,20.
15.e-6,400.

$$\varepsilon_{th} = \int_{120}^{T} \alpha(T) I_3 dT \quad \text{avec} \quad I_3 = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix}$$

Pratique des éléments finis

Dilation/contraction orthotrope: un CTE dans chacune des 3 directions.

*EXPANSION
Specify thermal or field expansion.

Data lines to define orthotropic thermal expansion coefficients (TYPE=ORTHO with USER parameter omitted):

First line:

1. α_{11}. (Units of θ^{-1}.)
2. α_{22}.
3. α_{33}. (Not used for plane stress and shell cases.)
4. Temperature.

***material, name=wood**
***expansion, zero = 120.**
******** CTE, temp**
22.e-6,5.e-6,5.e-6,20.
27.e-6,6.5.e-6,6.5.e-6,400.

$$\varepsilon_{th} = \int_{120}^{T} \begin{pmatrix} \alpha_{11}(T) & 0 & 0 \\ 0 & \alpha_{22}(T) & 0 \\ 0 & 0 & \alpha_{33}(T) \end{pmatrix} dT$$

Pratique des éléments finis

Les déformations thermiques peuvent mener à la plastification et à des contraintes résiduelles: calcul couplé thermique et mécanique.

***COUPLED TEMPERATURE-DISPLACEMENT**
Fully coupled, simultaneous heat transfer and stress analysis.

Optional parameters to control time incrementation in transient analysis:

DELTMX

Set this parameter equal to the maximum temperature change allowed within an increment. Abaqus/Standard will restrict the time step to ensure that this value is not exceeded at any node during any increment of the step. If both this and the CETOL parameter are omitted in a transient analysis, fixed time increments will be used, with a constant time increment equal to the initial time increment.

$$\text{DELTMX} = \underset{\text{all nodes}}{\overset{\text{over } \Delta t}{\text{Max}}} |\Delta T| = \text{variation max de } |\Delta T| \text{ sur le pas de temps } \Delta t$$

**** calcul sur 60 secondes
*Coupled Temperature-displacement, creep=none, deltmx=1.
****dt0, step time, min, max
0.1, 60., 0.0006, 5.

Pratique des éléments finis

Couplage thermomécanique: 4 degrés de libertés (NT + ux, uy et uz)
Eléments possibles: C3D8T, C3D20T, C3D20RT etc ….

Couplage faible: la température influence la mécanique via :
- l'expansion thermique
- et les prop. matériaux

Couplage fort: la mécanique influence en retour la thermique via :
- le transfert thermique dans les contacts entre pièces (ouverture/fermeture d'un gap, lame d'air …)
- la chaleur dissipée par déformation plastique …..

***GAP CONDUCTANCE**
Introduce heat conductance between interface surfaces.

```
*INELASTIC HEAT FRACTION
****************   fraction of inelastic dissipation rate that appears
****************  as a heat flux per unit volume
1.0,
```

Pratique des éléments finis

***GAP CONDUCTANCE**
Introduce heat conductance between interface surfaces.

$q = k\left(\theta_A - \theta_B\right)$, k = conductance en W/m2K (q est en W/m2)

When defining *k* directly, define it as

$$k = k(d, p, \bar{\theta}, \overline{|\dot{m}|}, \bar{f}_\gamma),$$

where

d

is the clearance between *A* and *B*,

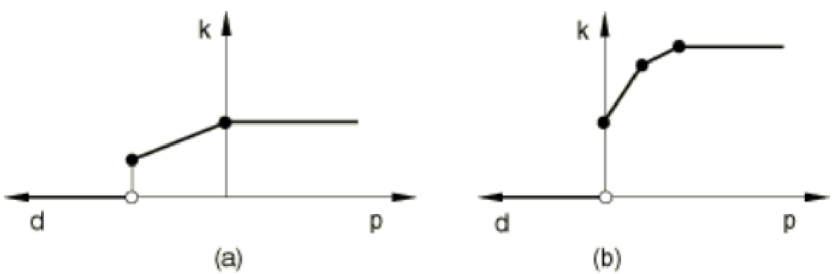
p

is the contact pressure transmitted across the interface between *A* and *B*,

$\bar{\theta} = \frac{1}{2}(\theta_A + \theta_B)$

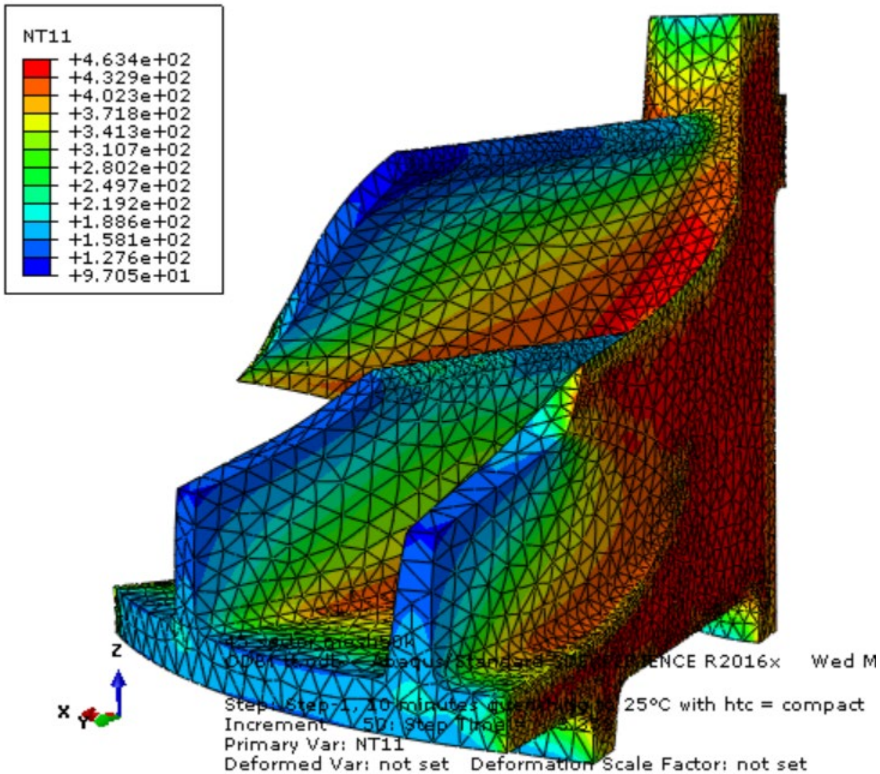
is the average of the surface temperatures at *A* and *B*,

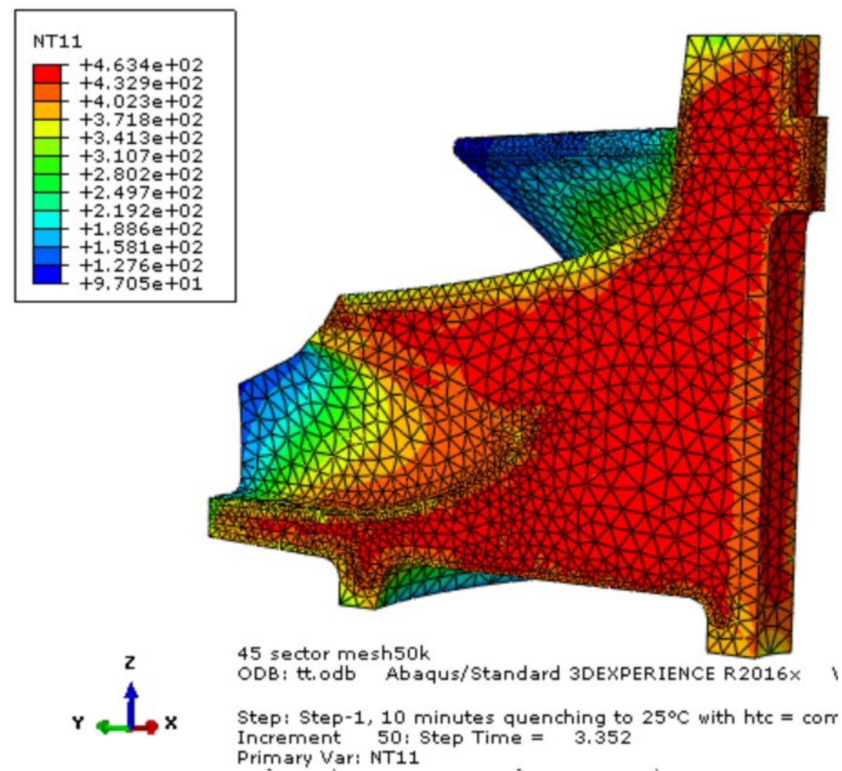
Figure 37.2.1–1 Examples of input data to define the gap conductance as a function of clearance or contact pressure.



Pratique des éléments finis.

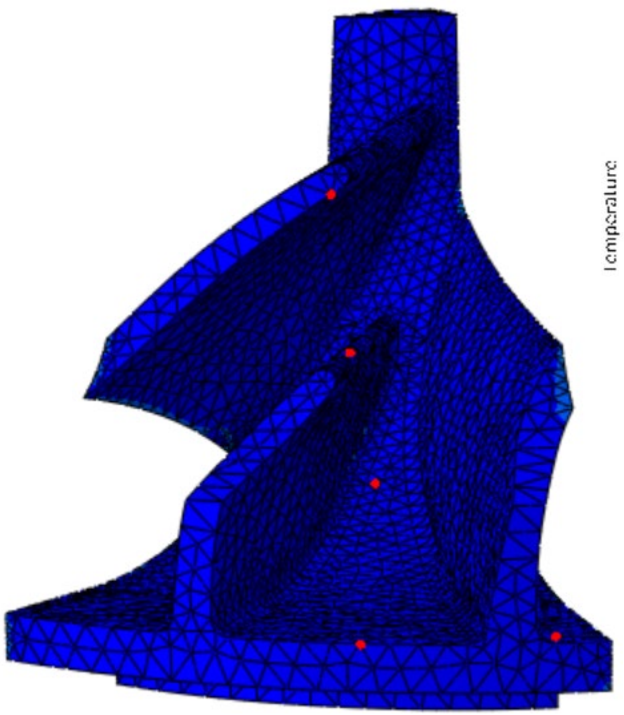
Calcul des contraintes résiduelles de trempe dans une aube de turbine.
Puis usinage de peau.
Relaxation des contraintes par spinning après trempe ……..

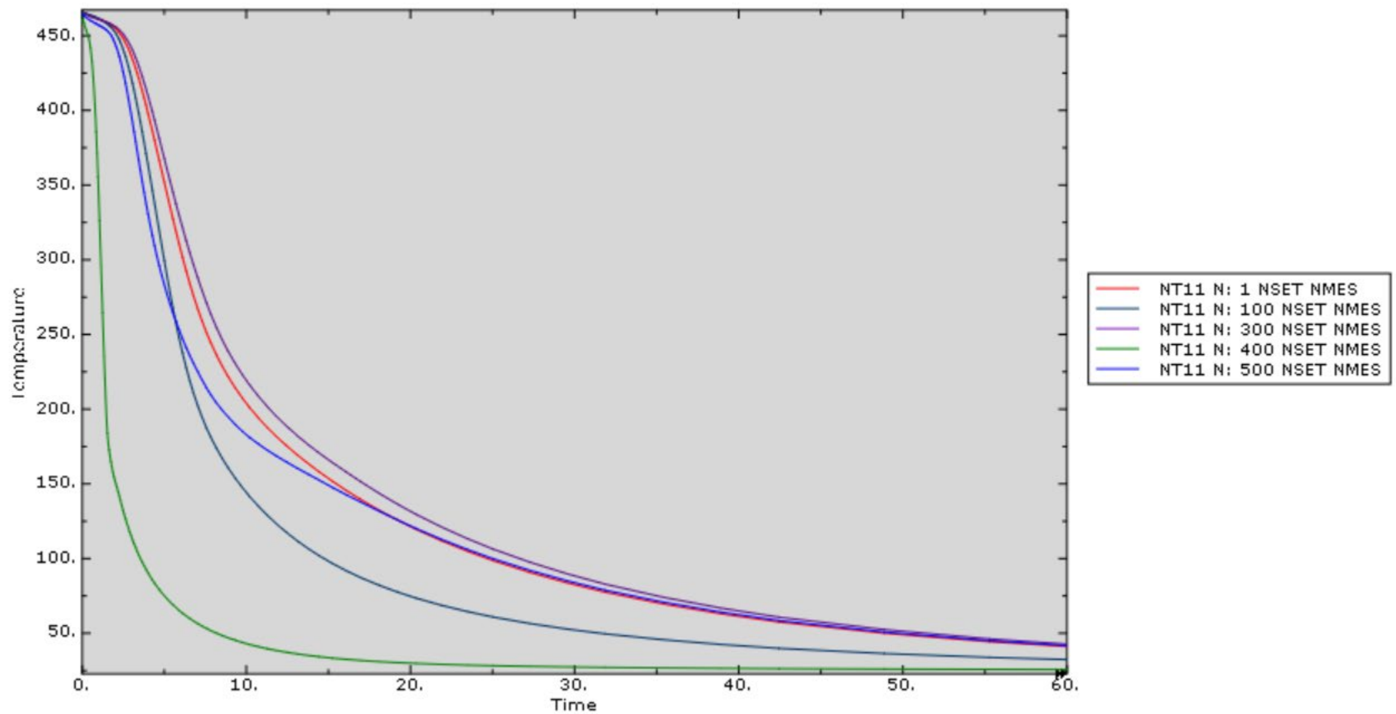




Pratique des éléments finis

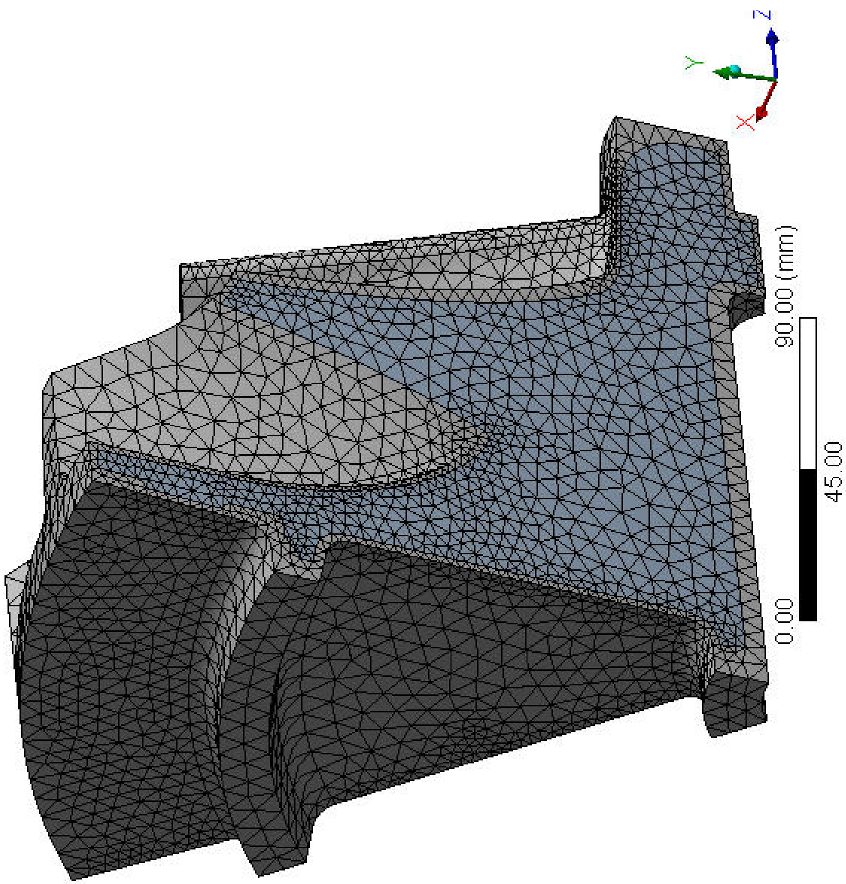
Calcul des contraintes résiduelles de trempe dans une aube de turbine.
Puis usinage de peau.
Relaxation des contraintes par spinning après trempe ……..

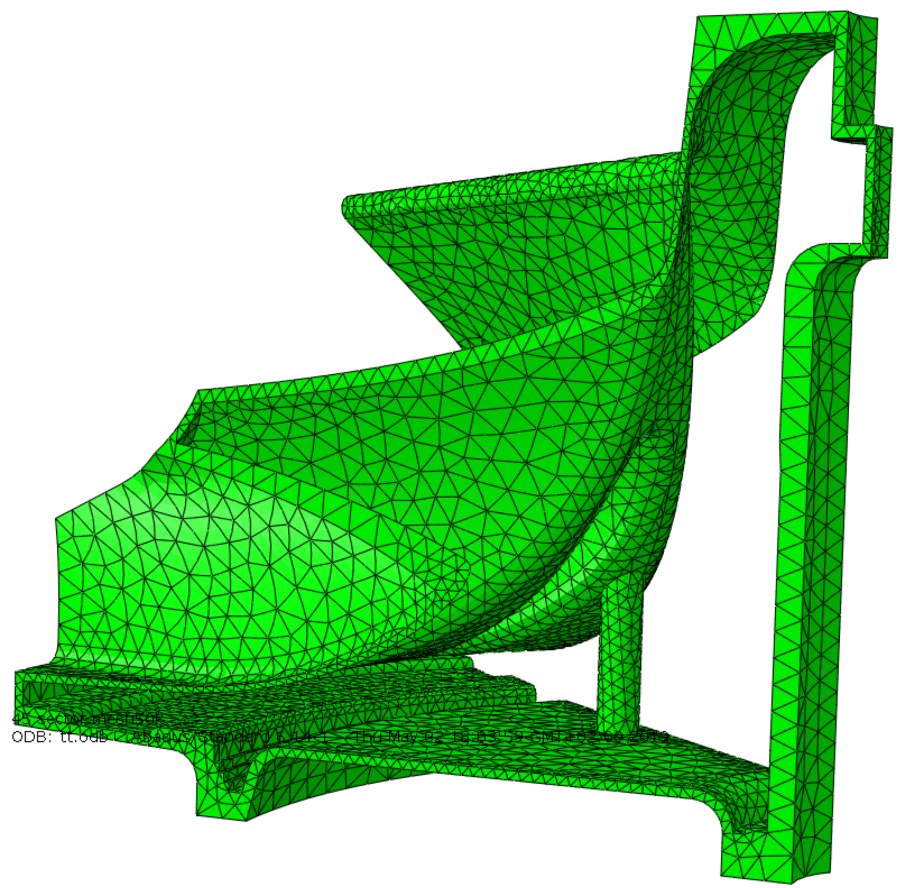




Pratique des éléments finis

Calcul des contraintes résiduelles de trempe dans une aube de turbine.
Puis usinage de peau (*model change, remove ….)
***model change, remove**
epeau





Pratique des éléments finis

Le fichier .msg contient les détails de la convergence et les éventuelles erreurs dues à une non-convergence.

```
STEP    1      INCREMENT     1     STEP TIME    0.00


                  S T E P      1     V I S C O   A N A L Y S I S


         loading

     AUTOMATIC TIME CONTROL WITH -
          A SUGGESTED INITIAL TIME INCREMENT OF                1.000E-03
          AND A TOTAL TIME PERIOD OF                           3.600E+04
          THE MINIMUM TIME INCREMENT ALLOWED IS                1.000E-06
          THE MAXIMUM TIME INCREMENT ALLOWED IS                1.000E+03

     THE SIZE OF THE TIME INCREMENT IS CONTROLLED BY -
          THE PARAMETER  CETOL                                 1.000E-04

          THE STEP WILL USE EXPLICIT TIME INTEGRATION,
          SWITCHING TO IMPLICIT INTEGRATION IF THIS
SEEMS NECESSARY.

 CONVERGENCE TOLERANCE PARAMETERS FOR FORCE
     CRITERION FOR RESIDUAL FORCE     FOR A NONLINEAR PROBLEM          5.000E-03
     CRITERION FOR DISP.    CORRECTION IN A NONLINEAR PROBLEM          1.000E-02
     INITIAL VALUE OF TIME AVERAGE FORCE                               1.000E-02
     AVERAGE FORCE     IS TIME AVERAGE FORCE
     ALTERNATE CRIT. FOR RESIDUAL FORCE     FOR A NONLINEAR PROBLEM    2.000E-02
     CRITERION FOR ZERO FORCE     RELATIVE TO TIME AVRG. FORCE         1.000E-05
     CRITERION FOR RESIDUAL FORCE     WHEN THERE IS ZERO FLUX          1.000E-05
     CRITERION FOR DISP.    CORRECTION WHEN THERE IS ZERO FLUX         1.000E-03
     CRITERION FOR RESIDUAL FORCE     FOR A LINEAR INCREMENT           1.000E-08
```

Pratique des éléments finis

le fichier .msg

La première itération ne converge pas ….

```
                CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1


AVERAGE FORCE                        2.38       TIME AVG. FORCE        2.38
LARGEST RESIDUAL FORCE              -11.1       AT NODE          148   DOF  3
LARGEST INCREMENT OF DISP.          -2.02       AT NODE        10045   DOF  2
LARGEST CORRECTION TO DISP.         -2.02       AT NODE        10045   DOF  2
         FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.
```

Après 4 itérations, le calcul converge: le résidu en force est très faible devant la force moyenne.

```
                CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     5


AVERAGE FORCE                        2.41       TIME AVG. FORCE        2.41
LARGEST RESIDUAL FORCE             -1.419E-06   AT NODE          206   DOF  2
LARGEST INCREMENT OF DISP.          -2.10       AT NODE        10045   DOF  2
LARGEST CORRECTION TO DISP.        -9.198E-06   AT NODE        10002   DOF  2
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED
```

Pratique des éléments finis

les résultats dans le fichier .msg

le premier incrément est enregistré dans le *.sta et le calcul se poursuit …..

Le second incrément ne converge pas après 2 itérations de Newton-Raphson: abaqus diminue alors le pas de temps de 0.001 s à 0.00025

STEP	INC	ATT	SEVERE DISCON ITERS	EQUIL ITERS	TOTAL ITERS	TOTAL TIME/ FREQ	STEP TIME/LPF	INC OF TIME/LPF	DOF MONITOR	IF RIKS
1	1	1	0	5	5	0.00100	0.00100	0.001000	-2.10	
1	2	1U	0	2	2	0.00100	0.00100	0.001000	-2.10	
1	2	2	0	3	3	0.00125	0.00125	0.0002500	-2.10	
1	3	1	0	1	1	0.00150	0.00150	0.0002500	-2.10	
1	4	1	0	1	1	0.00200	0.00200	0.0005000	-2.10	
1	5	1	0	1	1	0.00300	0.00300	0.001000	-2.10	

Pratique des éléments finis.

Convergence et consistance: h and p parameters

Un modèle EF est basé sur 2 sortes d'approximation:

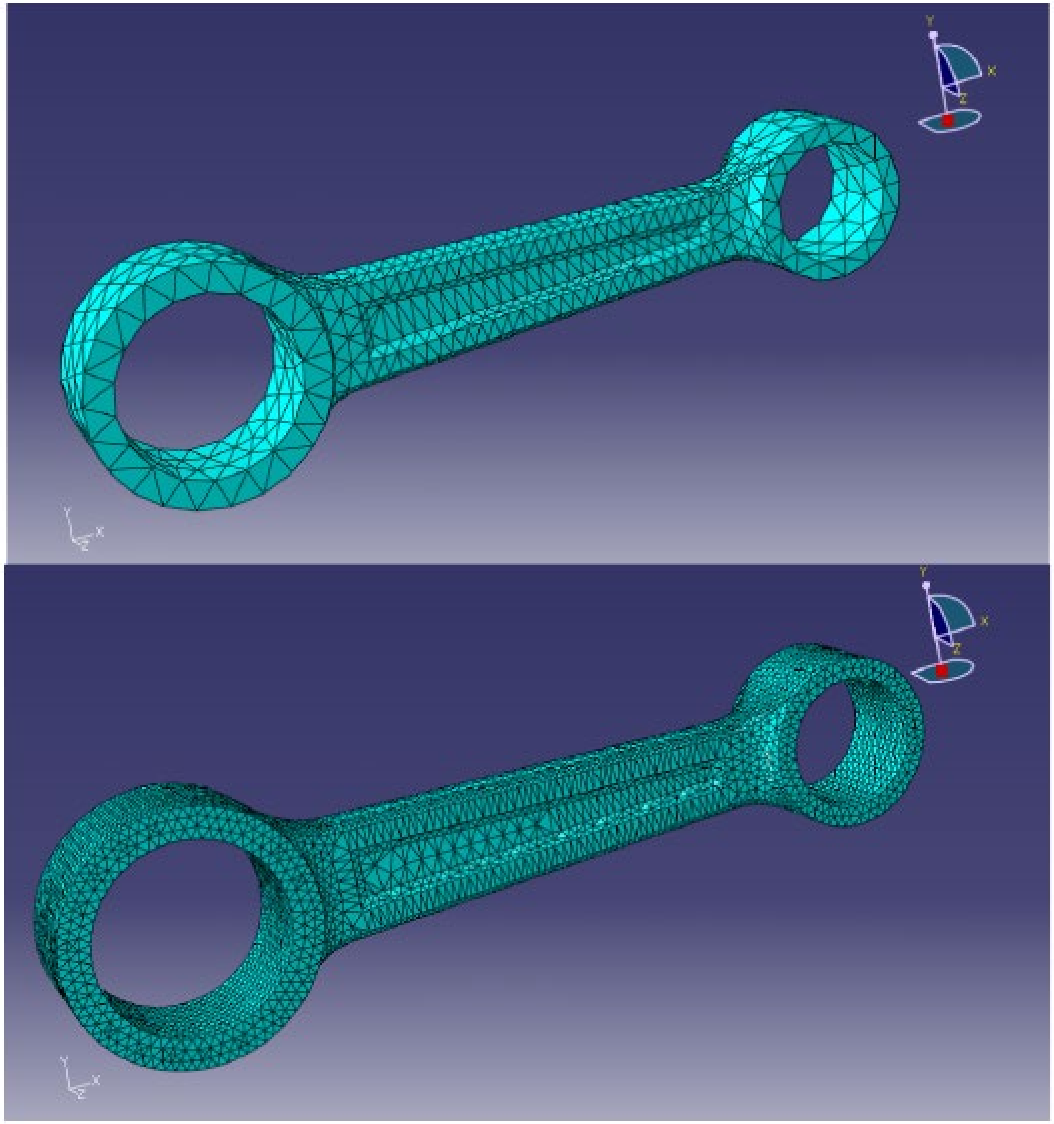
- Geometric discretization of the domain:
  - How fine the domain is discretized ?
    $\Rightarrow$ nb of elements
  - Determined by the characteristic size of the elements
  - This is called the h-parameter or h-refinement of the model.

- Internal approximation in each FE:
  - How accurate is the interpolation inside each element?
  - Determined by the order of the shape functions
  - This is called the p-parameter of the model.
  - Changes the number of nodes of the elements.

The main question is then:
how can we choose the ”right” element size h and FE formulation order p for a given problem ?

Pratique des éléments finis.

h-refinement: decrease global mesh size



$h = 5$

1421 nodes
5132 elements

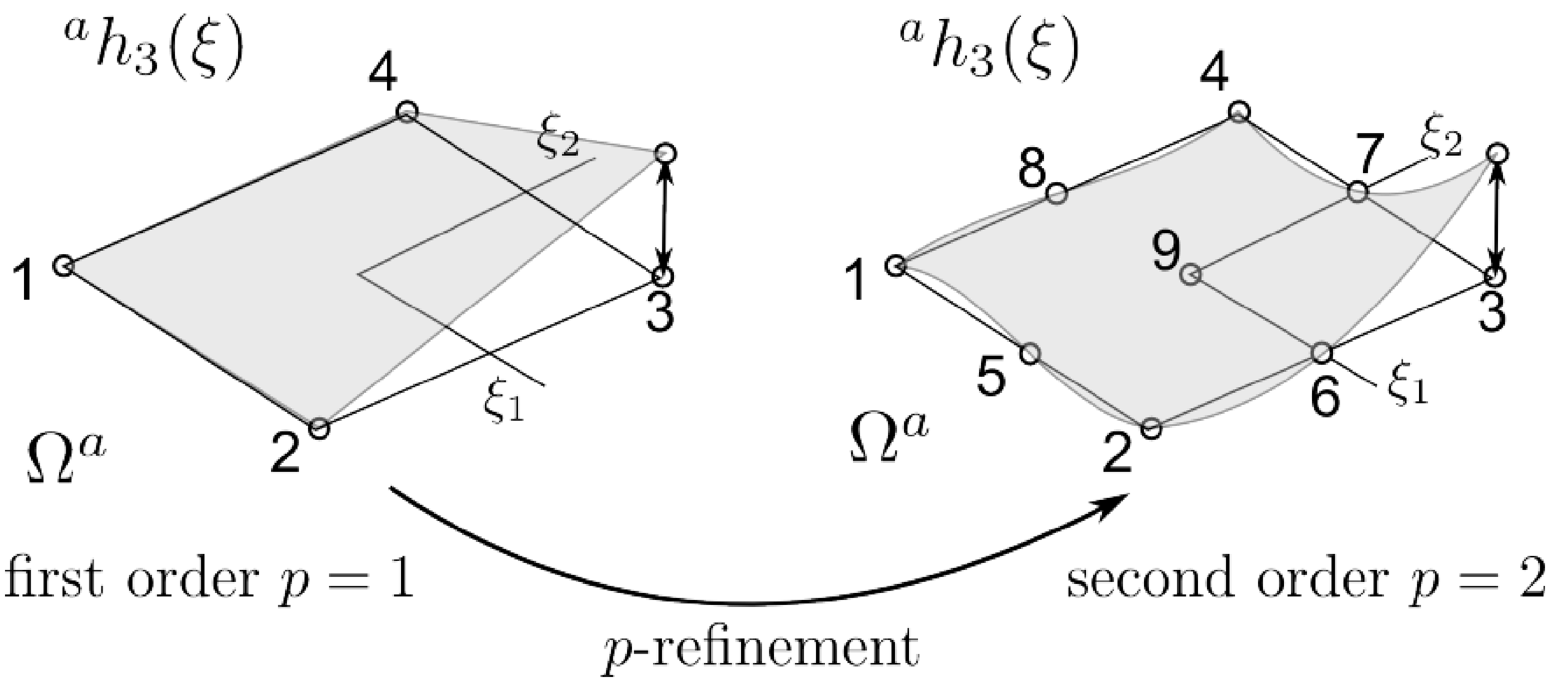
h-refinement

$h = 1.5$

8825 nodes
37913 elements

Pratique des éléments finis.

p-refinement for 2D elements: linear (4 nodes) to quadratic elements (9 nodes)



Pratique des éléments finis.

Précision (accuracy) et nombre de degrés de liberté NDOF:

Altogether, the h and p parameters define the total number of degrees of freedom of the model:

- if h decreases (finer mesh), with have more elements and thus more nodes (each node as N DOFs)$\Rightarrow$ increase of the total number of DOFs.
- if p increases, the number of nodes in each element increases $\Rightarrow$ increase of the total number of DOFs.
- in many application, the important criteria is *accuracy* as a function of *computationnal cost*.
- computational cost of a model is depending approx. on $O(ndof^3)$
- thus in practice, the optimal *accuracy / ndof trade-off* is the main criteria used to select the (h,p) parameters of a model.

Convergence:

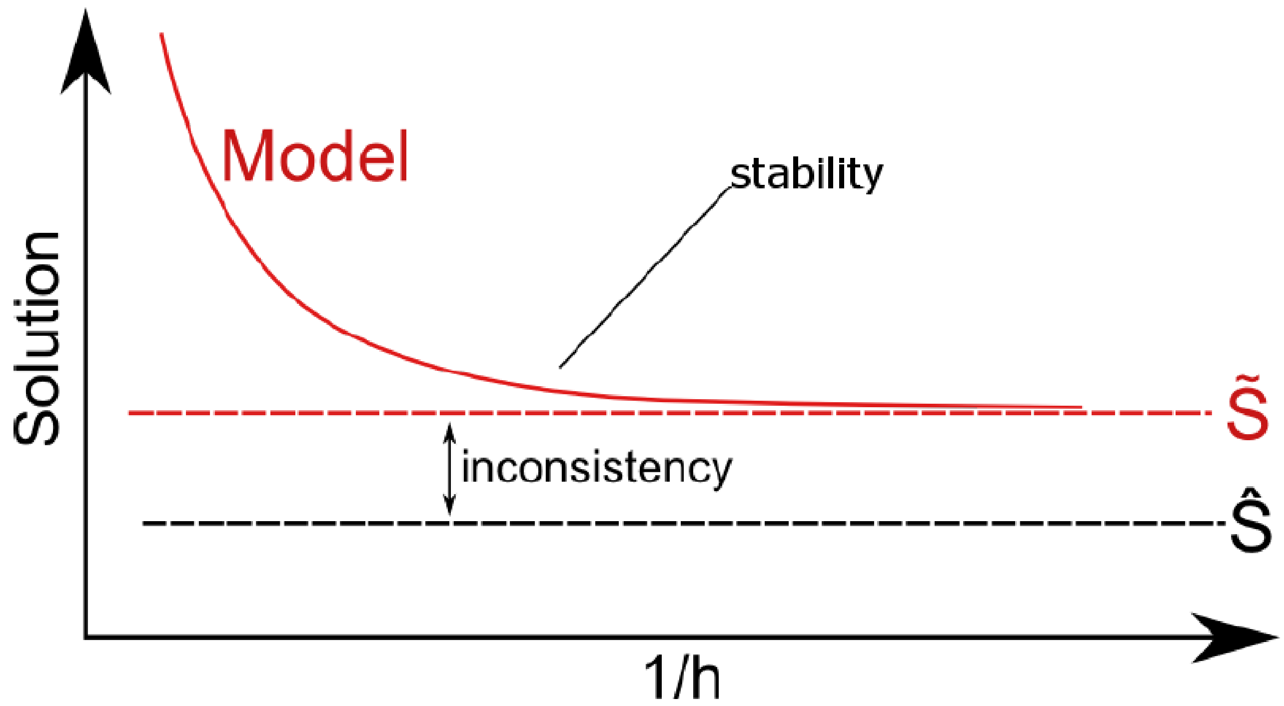
An approximate model *converges* with respect to one of its approximation parameters (h or p) if its solution $S_{h,p}$ tends to the exact solution $\hat{S}$ when $h \to 0$ and / or $p \to \infty$. Convergence is ensured if the model is *consistent* and *stable*.

Stabilité et consistance:

If the numerical formulation is *stable*, the solution $S_{h,p}$ usually tends to a limit $\tilde{S}$ (stagnation of the solution).
However, this does not mean that the "limit" $\tilde{S}$ is the exact solution $\hat{S}$.
If the numerical model is also *consistent*, we have $\tilde{S} = \hat{S}$,which means that the numerical approximation *solves exactly the same problem* as the analytical formulation (*consistency*).

Pratique des éléments finis.

Pour p donné et h variable :



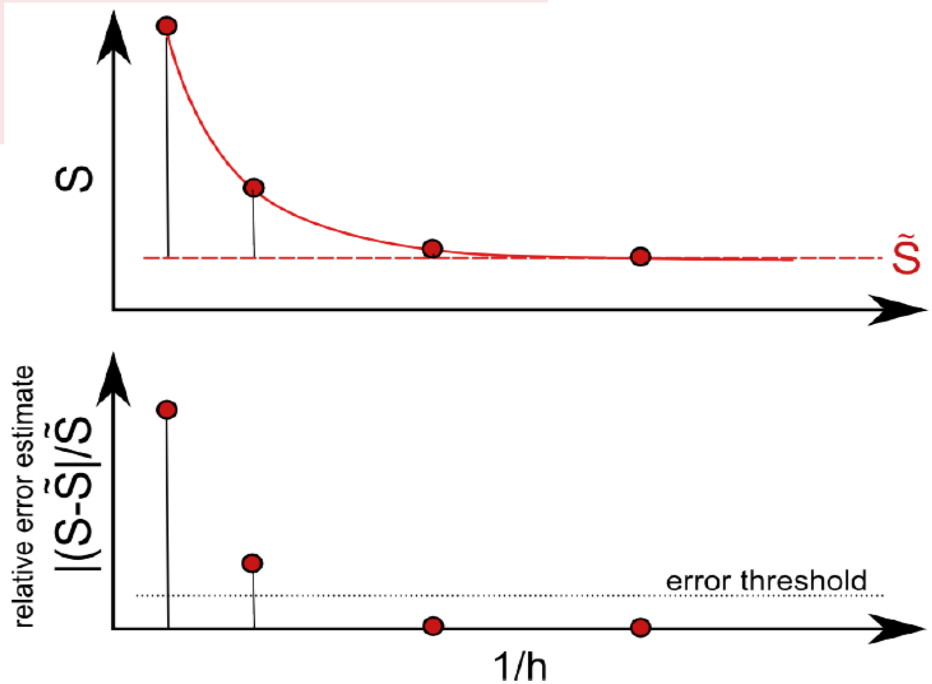
In practice, most finite element formulations are consistent so this point is not checked systematically in every study.

However a mesh convergence study on the h parameter MUST always be done.

Pratique des éléments finis.

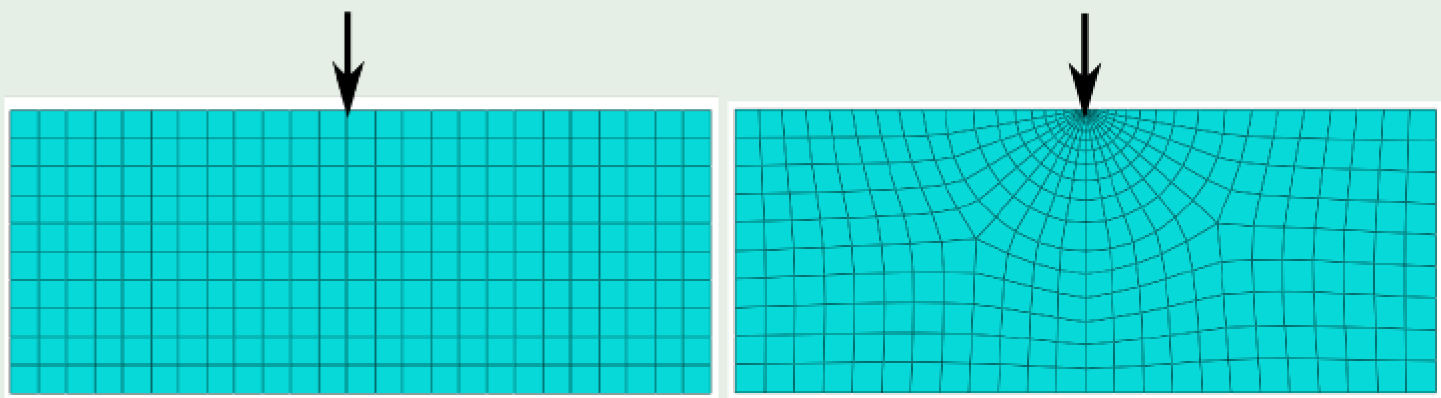
Etude de convergence :

- vary h in your model, choose at least 3 values of h. Choose the smallest h such that it corresponds to the finest model that you can run within an acceptable time.
- take the finest model as a reference, let's call its solution $\tilde{S}$
- analyze the evolution of the relative error $RE_{h,p} = \| (S_{h,p} - \tilde{S})/\tilde{S} \|$:
- when the relative error RE is less than a chosen error threshold (which depends on your precision requirement), the mesh is sufficiently refined.
- If not, continue to refine the mesh



Affinage local : utilisation de bias et autres possibilités

When a local value is of interest, like for example the maximum von Mises stress in a stress concentration area, a local mesh refinement ($h_{local} < h_{global}$) is highly recommended. Similarly to the global mesh convergence analysis on h_{global}, it is recommended to also check that the local target value converges with the local mesh size h_{local}.



Pratique des éléments finis

Abaqus cae: tapez abaqus cae
Nombreux tutorials sur youtube …

https://www.youtube.com/watch?v=XvHaVep-VKs

Propé2 : mercredi 18 Décembre 24, 15h15 à 18h
Accès à moodle
Cas sur Abaqus
Résultats sur une feuille individuelle rendue

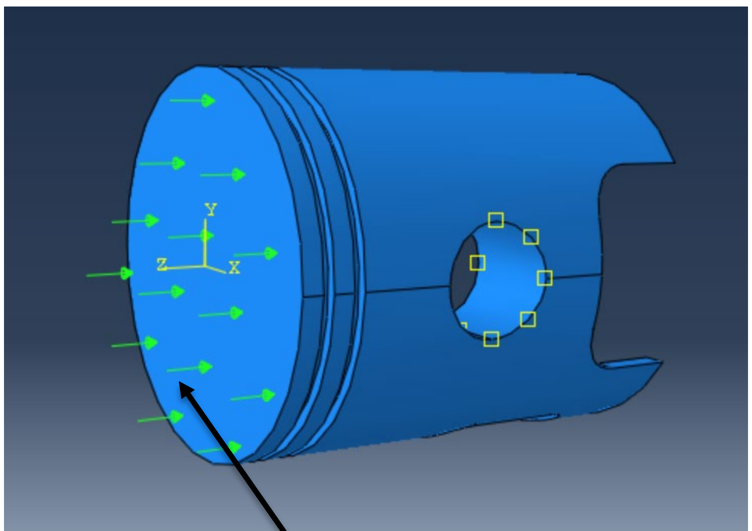
Noémie et moi-même répondrons à vos questions d'ordre informatique (logiciels, fichiers, …)

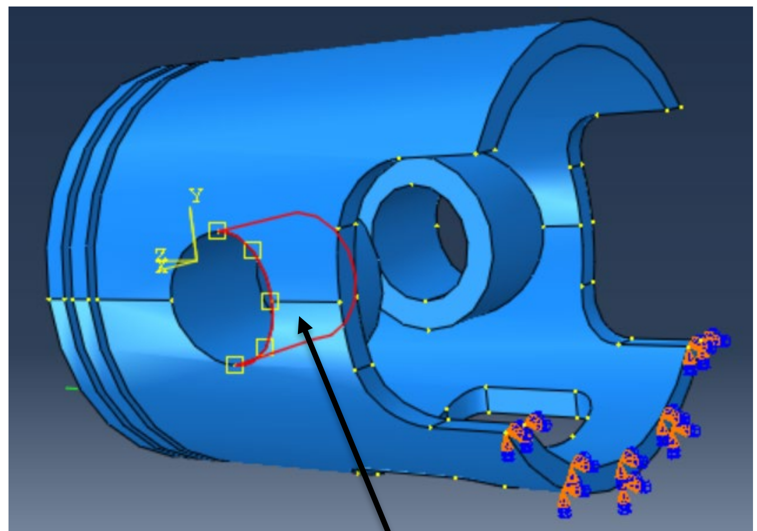
Pratique des éléments finis

Thermo-mécanique dans un piston
(tm-piston.inp et piston-mesh.dat)

Maillage: *include, file=piston-mesh.dat
Initialement le piston est à 20°C, thermique transitoire sur 2h.
Nœuds encastrés sur la demi face maintenue à 20°C
Chauffage: flux de 2.10^5 W/m2 sur la surface supérieure pendant 1 h
Refroidissement: flux de 0.0 pendant 1 heure.
(temps de calcul = 5 heures !!!)

```
*Material, name=alu
*Conductivity
 0.2,
*Density
 2.7e-06,
*Elastic
70000., 0.3
*Plastic
150.,0.,20.
170., 0.02,20.
200., 0.05,20.
300.,  0.1,20.
50.,0.,420.
70., 0.02,420.
100., 0.05,420.
200.,  0.1,420.
*Expansion
 3e-05,
*Specific Heat
420.
```





Flux de chaleur

Encastré et maintenu à 20°C

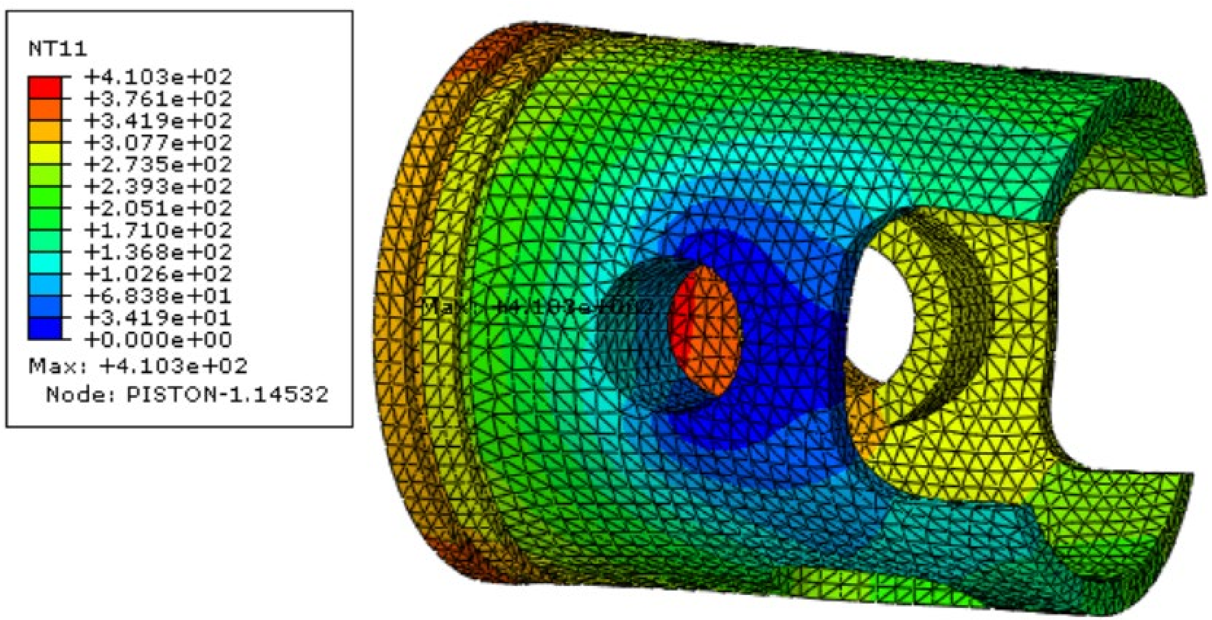
Pratique des éléments finis

Thermo-mécanique dans un piston

Que vaut Von Mises max à l'issue de la montée en température ?
Puis après le refroidissement ? (affiner le maillage localement …)

Le matériau a plastifié lors du chauffage proche des endroits encastrés.

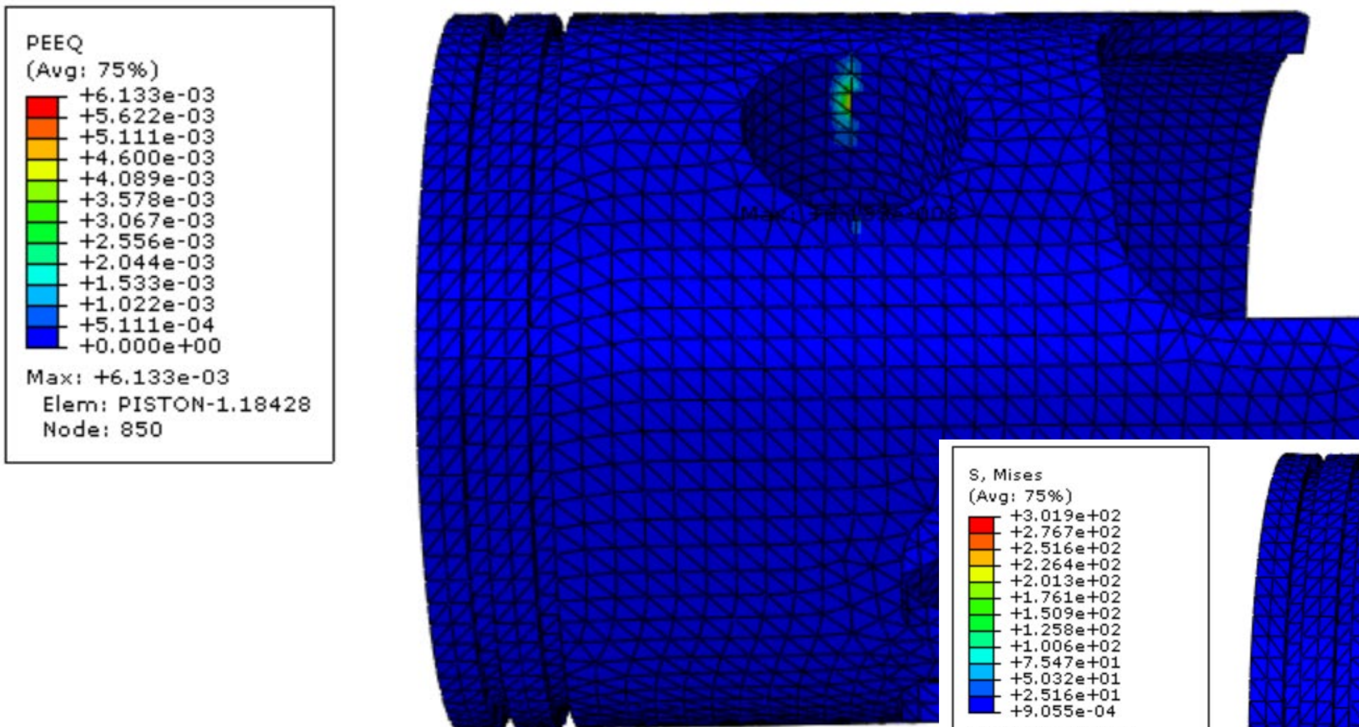
Et des contraintes résiduelles sont apparues.

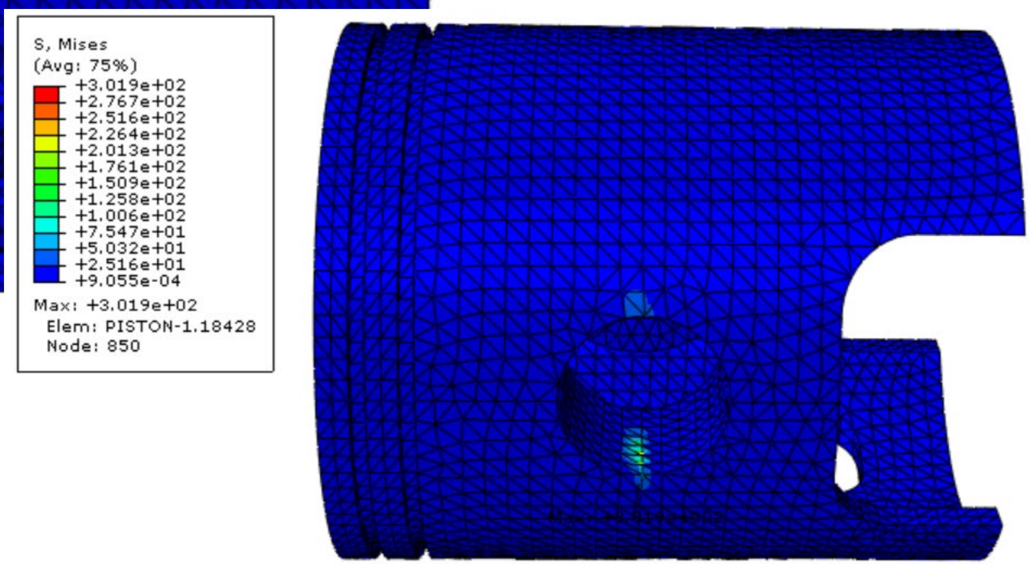


Pratique des éléments finis

Thermo-mécanique dans un piston

La pièce plastifie légèrement à l'endroit de l'encastrement dans le trou: il apparait alors de contraintes résiduelles

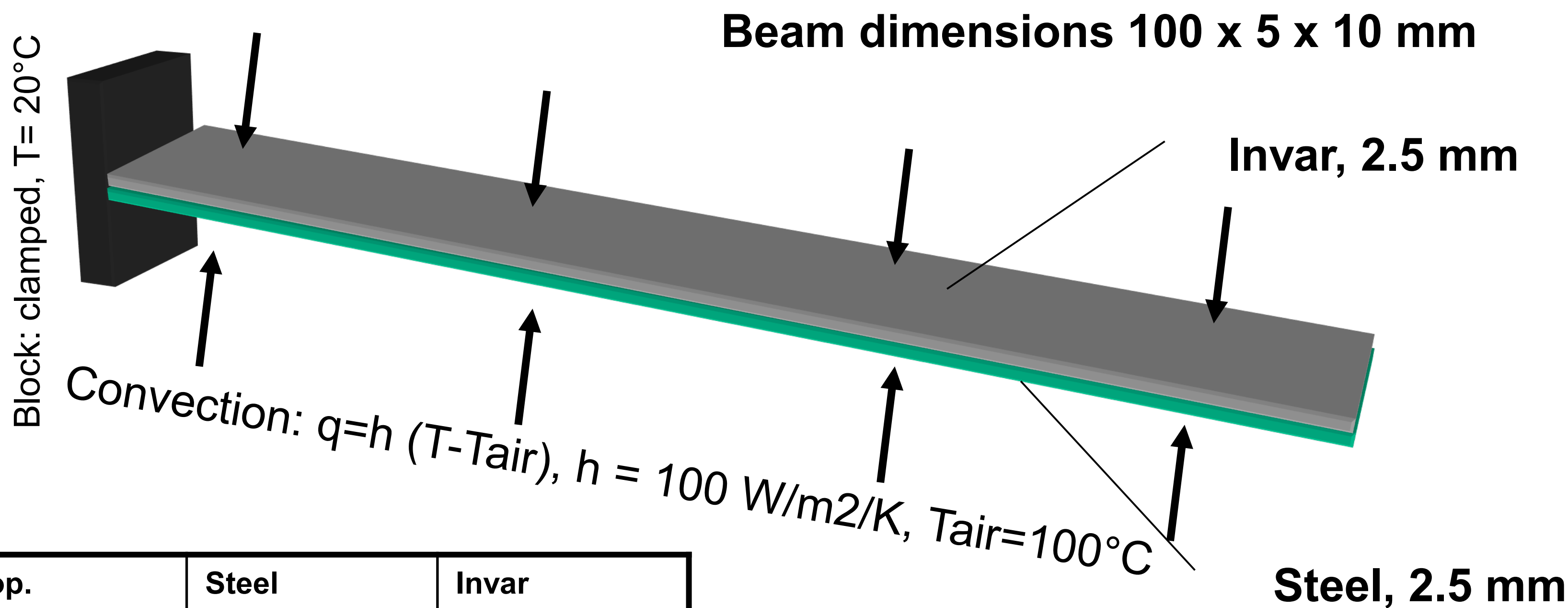




Pratique des éléments finis.

Exo 11: bilame soumis à un chargement thermique

En partant du fichier poutre-lf.inp, modifiez le maillage et définissez 2 matériaux pour simuler un bilame avec pour unités le système N, mm, kg, J et s. Calculer la flèche du bilame à 100 sec.



Prop.	Steel	Invar
Young's modulus	210 GPa	141 GPa
Poisson ratio	0.3	0.3
Th. Expansion	1e-5 /K	1e-6 /K
Density	7800 kg/m3	8000 kg/m3
Conductivity	30 W/m/K	10 W/m/K
Specific heat	1000 J/kg/K	500 J/kg/K

Le chauffage sur les parties supérieures et inférieures se fait à l'aide de
*sflim
……