

Mécanique des solides

1. Modéliser le comportement des matériaux

Reine Fares

reine.fares@cea.fr

Cours de Modélisation par Abaqus

- Comportement cyclique des matériaux
 - Élément cubique unitaire pour tester les lois de comportement
- Poutre 1D appuyée et encastrée
 - Données : géométrie, charges, matériau, conditions aux limites
 - Résultats : flèche, contrainte max, réactions
- Déformées modales d'une structure
 - Premiers modes de vibration et masses associées
- Réservoir d'eau en décélération
 - Données : géométrie, matériau, charges statiques et linéaire
 - Résultats : déplacement dans le temps, distribution vide/matériel
- Poutre 3D en béton armé
 - Géométrie 3D, éléments finis solides

Tenseurs de contrainte

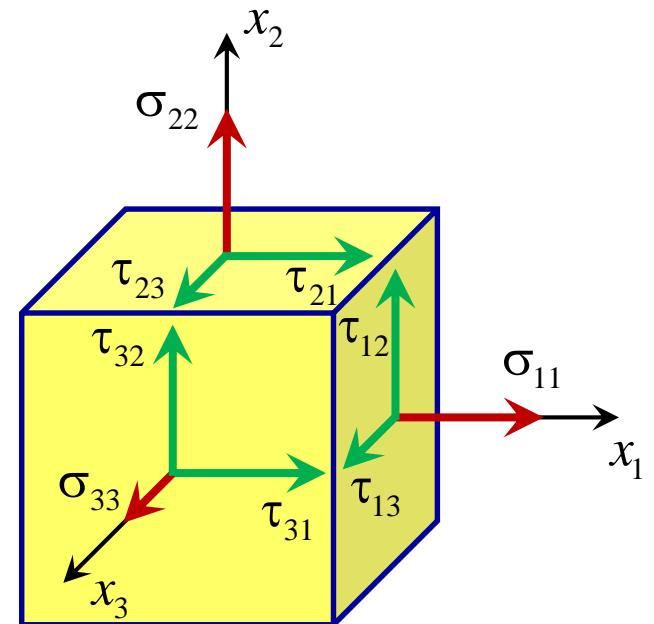
- Le tenseur de contrainte dépend du repère choisi
- Les contraintes dans les facettes positives sont positives selon la direction des axes
- Pression : contrainte négative
- **Tenseur de contrainte :** matrice symétrique $\tau_{ij} = \tau_{ji}$, unité de mesure [N/m²]

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \tau_{12} & \tau_{13} \\ \tau_{21} & \sigma_{22} & \tau_{23} \\ \tau_{31} & \tau_{32} & \sigma_{33} \end{bmatrix}$$

- **Vecteur de contrainte :**

$$\boldsymbol{\sigma} = [\sigma_{11} \quad \sigma_{22} \quad \sigma_{33} \quad | \quad \tau_{23} \quad \tau_{13} \quad \tau_{12}]^T$$

Normales **Tangentielles**



Tenseurs de déformation

- **Matrice de déformation :** matrice symétrique $\varepsilon_{ij} = \varepsilon_{ji}$
- **Vecteur de déformation**

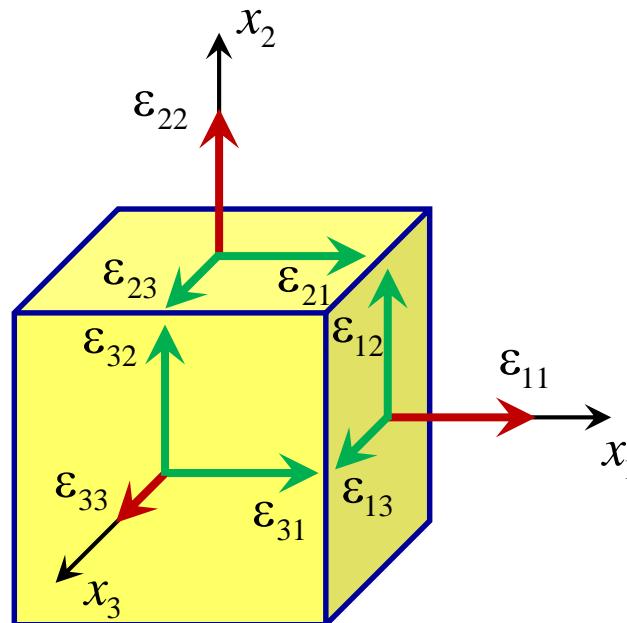
$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} & \varepsilon_{12} & \varepsilon_{13} \\ \varepsilon_{21} & \varepsilon_{22} & \varepsilon_{23} \\ \varepsilon_{31} & \varepsilon_{32} & \varepsilon_{33} \end{bmatrix}$$

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} & \varepsilon_{22} & \varepsilon_{33} & \mid & \gamma_{23} & \gamma_{13} & \gamma_{12} \end{bmatrix}^T$$

Axiales

De cisaillement

$$\gamma_{ij} = 2\varepsilon_{ij}$$

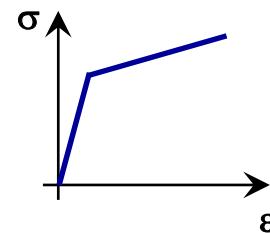
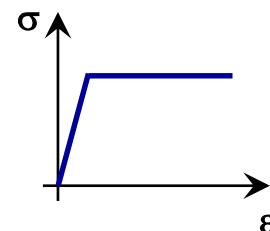
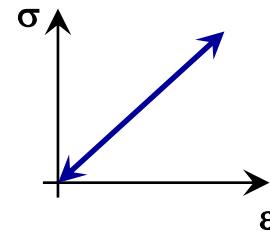


Types de comportement 1/4

- Modélisation du comportement d'un matériau : relation entre déformation et contrainte

- Élastique linéaire

$$\left. \begin{array}{l} \text{• Parfaitement plastique} \\ \text{• Plastique avec écrouissage} \end{array} \right\} \sigma = E \varepsilon$$



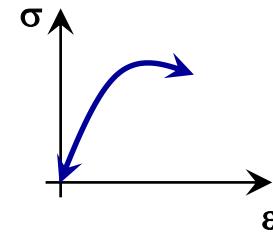
Types de comportement 2/4

- Modélisation du comportement d'un matériau : relation entre déformation et contrainte

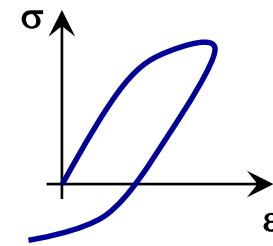
- Élastique non linéaire

}

$$\Delta\sigma = E \Delta\varepsilon$$



- Plastique non linéaire

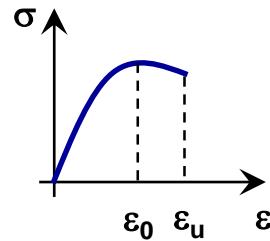


Types de comportement 3/4

- Comportement après la perte d'élasticité

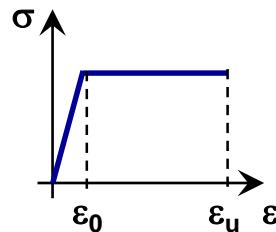
- **Matériau fragile** $\varepsilon_u \ll \varepsilon_0$

Béton, Verre, Roche, Fonte



- **Matériau ductile** $\varepsilon_u \gg \varepsilon_0$

Acier, Cuivre

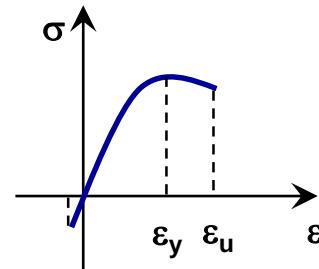


Types de comportement

- Comportement en traction et en compression

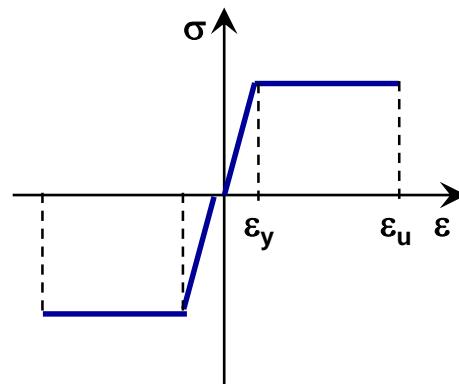
- Asymétrique et fragile

Béton, Roche



- Symétrique et ductile

Acier, Métaux



Matrice constitutive pour matériaux élastiques isotropes linéaires

- Paramètres à définir : \mathbf{E} , \mathbf{G} , ν

$$\boldsymbol{\sigma} = \mathbf{E} \boldsymbol{\varepsilon}$$

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \tau_{12} \\ \tau_{13} \\ \tau_{23} \end{bmatrix} = \begin{bmatrix} \frac{E(\nu-1)}{(\nu+1)(2\nu-1)} & \frac{-\nu E}{(\nu+1)(2\nu-1)} & \frac{-\nu E}{(\nu+1)(2\nu-1)} & 0 & 0 & 0 \\ \frac{-\nu E}{(\nu+1)(2\nu-1)} & \frac{E(\nu-1)}{(\nu+1)(2\nu-1)} & \frac{-\nu E}{(\nu+1)(2\nu-1)} & 0 & 0 & 0 \\ \frac{-\nu E}{(\nu+1)(2\nu-1)} & \frac{-\nu E}{(\nu+1)(2\nu-1)} & \frac{E(\nu-1)}{(\nu+1)(2\nu-1)} & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{bmatrix}$$

Loi de comportement 1/2

- **Données** : élément cubique unitaire pour tester les lois de comportement

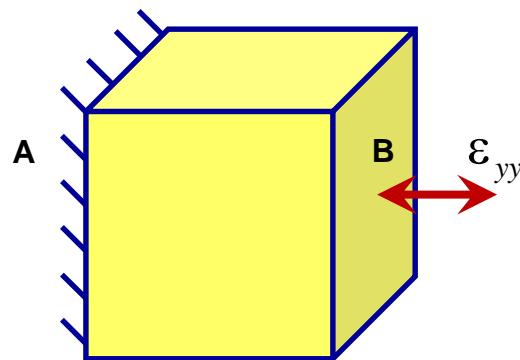
- Géométrie **3D** : $1 \times 1 \times 1$ m
- Matériau : acier (élastique-parfaitement plastique)

Module de Young $E = 210 \cdot 10^9$ N/m², Coefficient de Poisson $\nu = 0.3$, Densité $\rho = 8$ t/m³

Perte d'élasticité : $\varepsilon_y = 0.002$, $f_y = 450$ N/mm²

Rupture : $\varepsilon_u = 0.1$, $f_u = 540$ N/mm²

- Conditions aux limites : A) $u_y = 0$, axe de symétrie x : $u_z = 0$, axe de symétrie z : $u_x = 0$
- Contrôle en déformation : B) $u_y \neq 0$ (compression/traction)

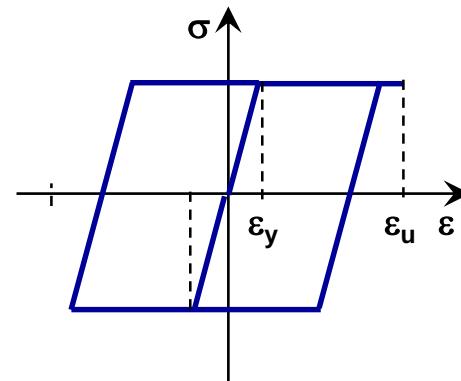
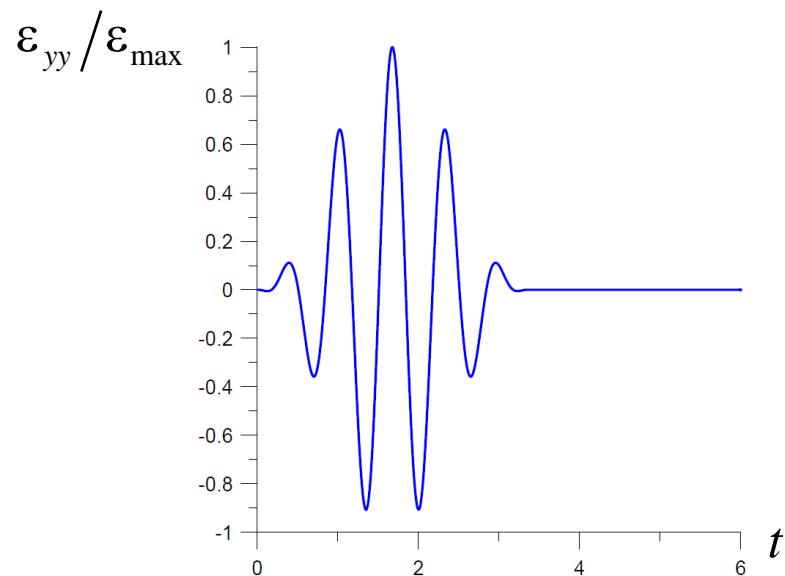
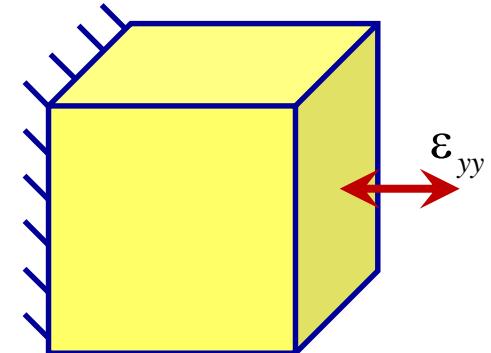


Loi de comportement 2/2

- **Résultats :**

- Courbe déformation-contrainte :

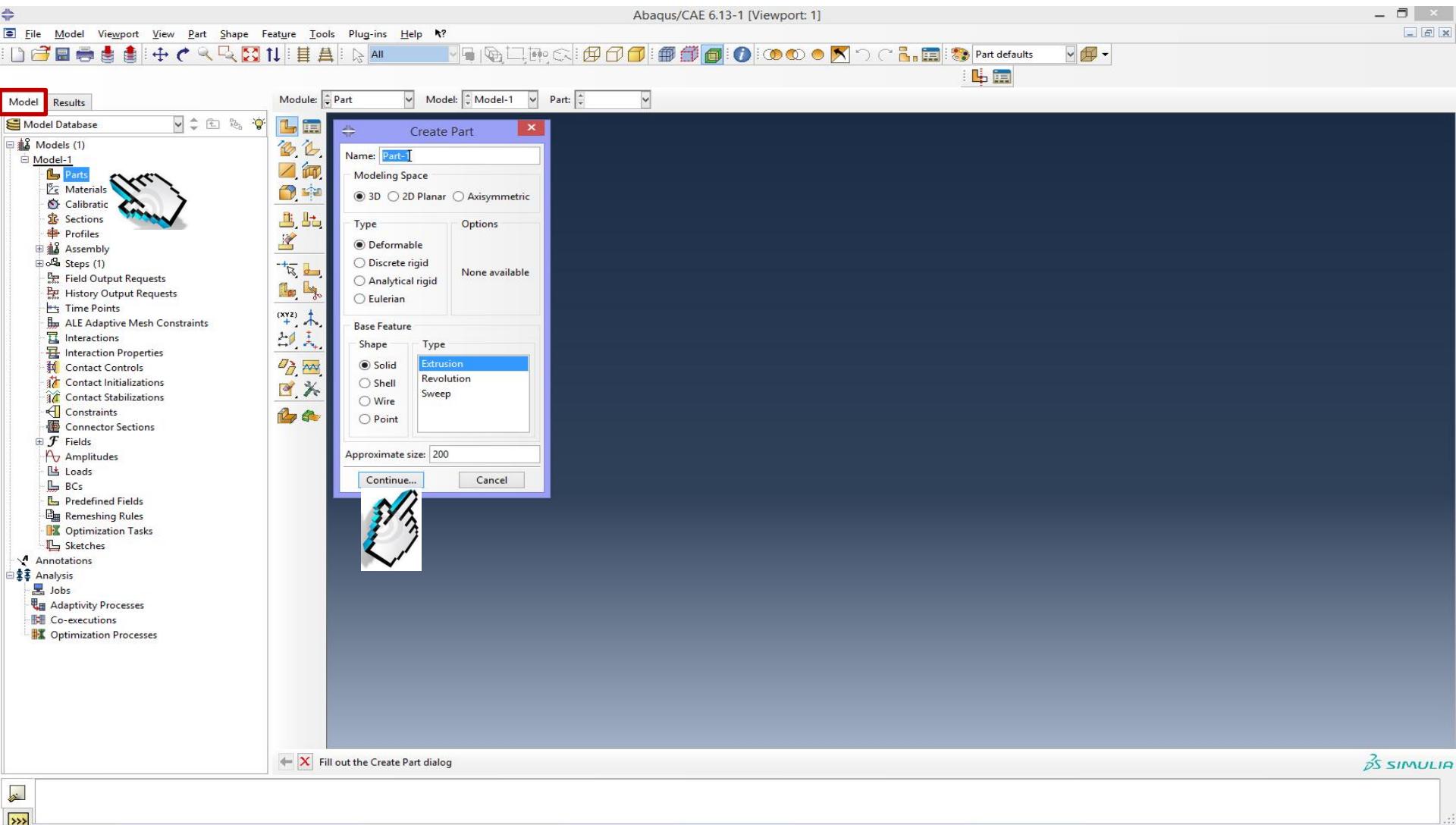
pour le cube unitaire déplacement \equiv contrainte



**Comportement hystérotique
sous charge cyclique
quasi-statique ou dynamique**

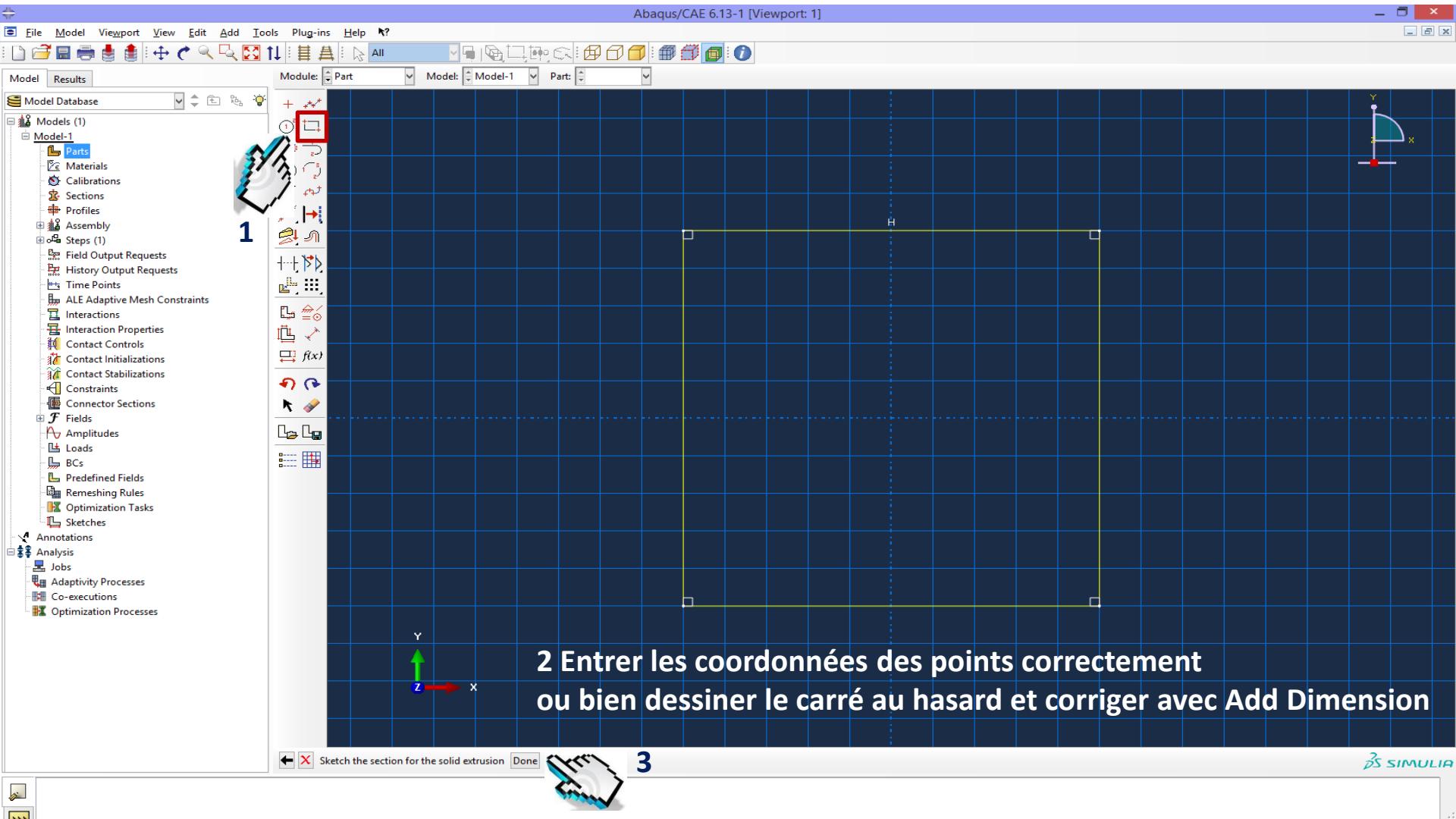
Model - Parts

1. Géométrie : type d'analyse (3D, 2D), type d'élément (solide, plaque, poutre)



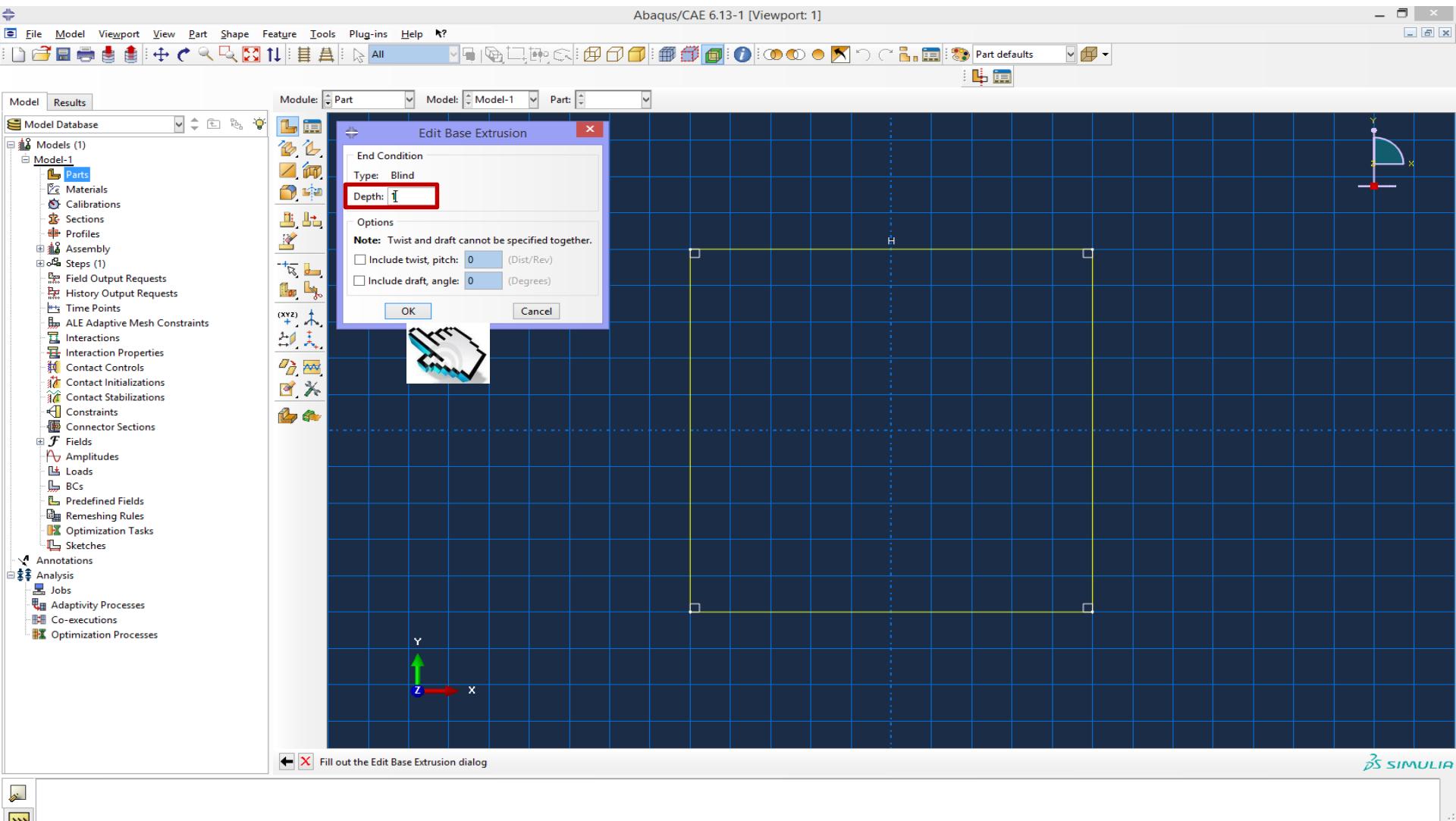
Model - Parts

1. Géométrie : section



Model - Parts

1. Géométrie : épaisseur



Model - Materials

2. Matériaux : densité

Abaqus/CAE 6.13-1 [Viewport: 1]

File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help ?

Module: Property Model: Model-1 Part: Part-1

Unités de mesure du SI : kg, m, s, N

Model Results Material Library

Model Database

Models (1) Model-1 Parts (1) Materials Calibrations Sections Profiles

Edit Material

Name: Acier Description:

Material Behaviors

Density

General Mechanical Thermal Electrical/Magnetic Other

Density Depvar Regularization User Material User Defined Field User Output Variables

Data

Mass Density 1 8000

OK Cancel

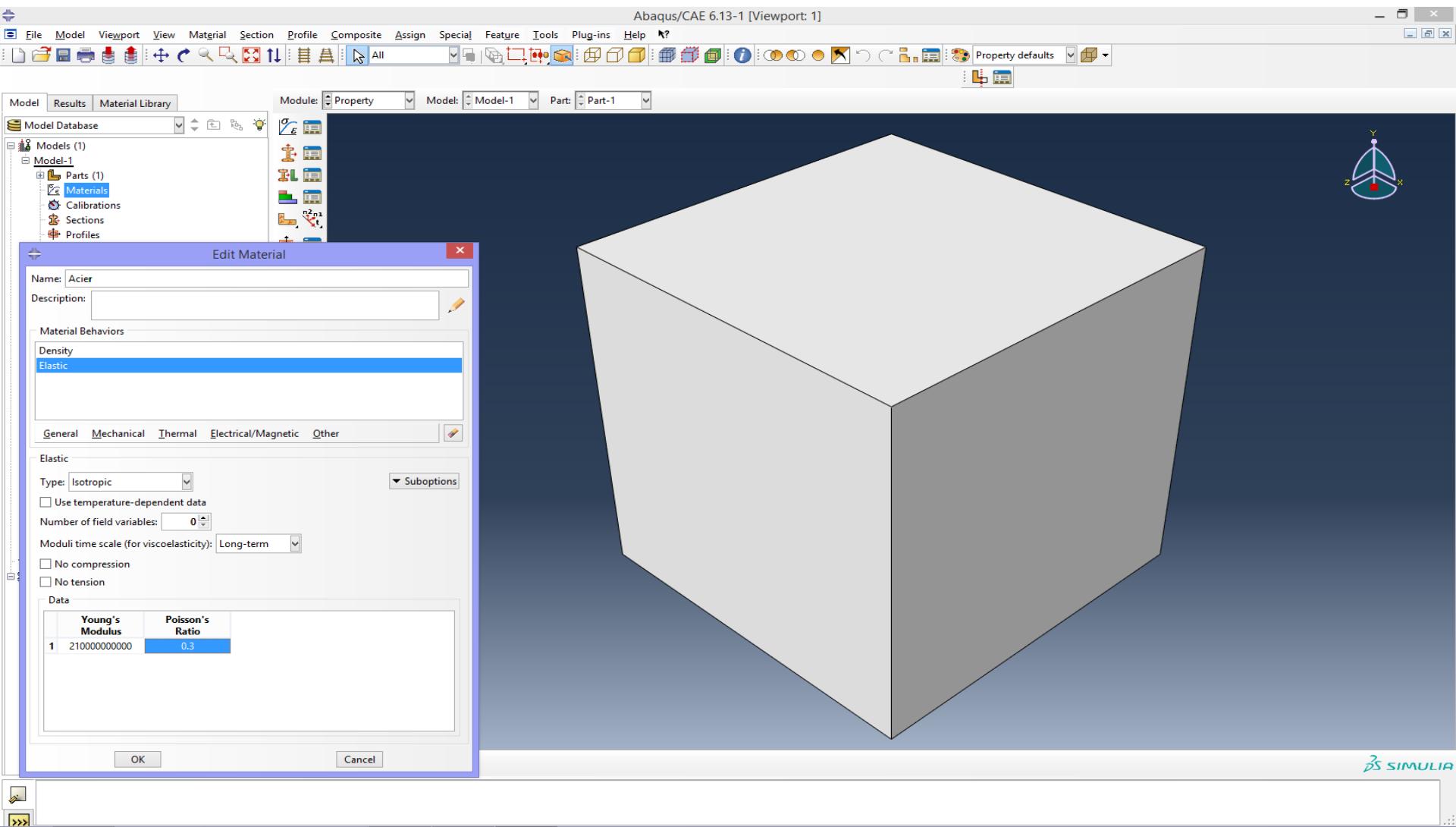
Property defaults

Y Z X

SIMULIA

Model - Materials

2. Matériaux : élasticité (E , ν)



Model - Materials

2. Matériau : plasticité (σ_y , ϵ_p)

Abaqus/CAE 6.13-1 - Model Database: C:\UNICE_teaching\Abaqus\Steel_element\element.cae [Viewport: 1]

File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help ?

Module: Property Model: Model-1 Part: Part-1

Pour le calcul numérique c'est mieux d'ajouter un peu d'écrouissage

Name: acier

Description:

Material Behaviors

Density

Elastic

Plastic

General Mechanical Thermal Electrical/Magnetic Other

Plastic

Hardening: Isotropic

Use strain-rate-dependent data

Use temperature-dependent data

Number of field variables: 0

Data

Yield Stress	Plastic Strain
1 45000000	0
2 54000000	0.098

$\sigma_y \quad \epsilon_p = \epsilon - \epsilon_y$

OK Cancel

Y X Z

σ

σ_y

ϵ_y

ϵ_u

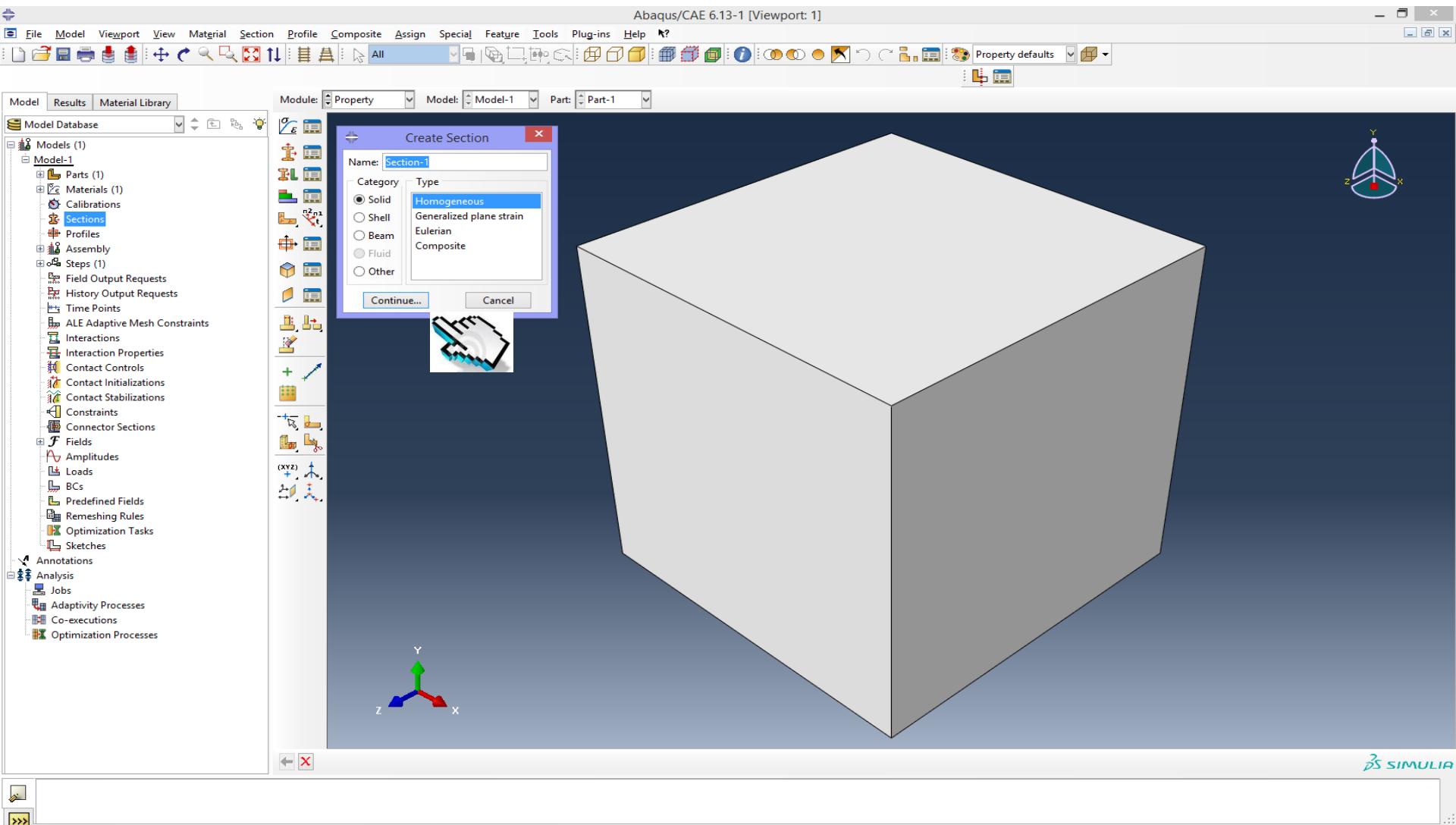
$\epsilon_p = \epsilon - \epsilon_y$

SIMULIA

The screenshot displays the Abaqus/CAE software interface. On the left, the 'Edit Material' dialog is open for a material named 'acier'. It shows a table with two entries: Yield Stress (45000000) and Plastic Strain (0.098). Below the table, the formula $\epsilon_p = \epsilon - \epsilon_y$ is displayed. On the right, a 3D model of a steel element is shown in green, and a stress-strain graph is overlaid. The graph illustrates an elastic-plastic material behavior with an isotropic hardening law. Key points marked on the graph are the yield stress σ_y , yield strain ϵ_y , ultimate stress ϵ_u , and plastic strain $\epsilon_p = \epsilon - \epsilon_y$. The graph is bounded by dashed lines. The Abaqus interface includes a toolbar, a menu bar, and a property manager at the top.

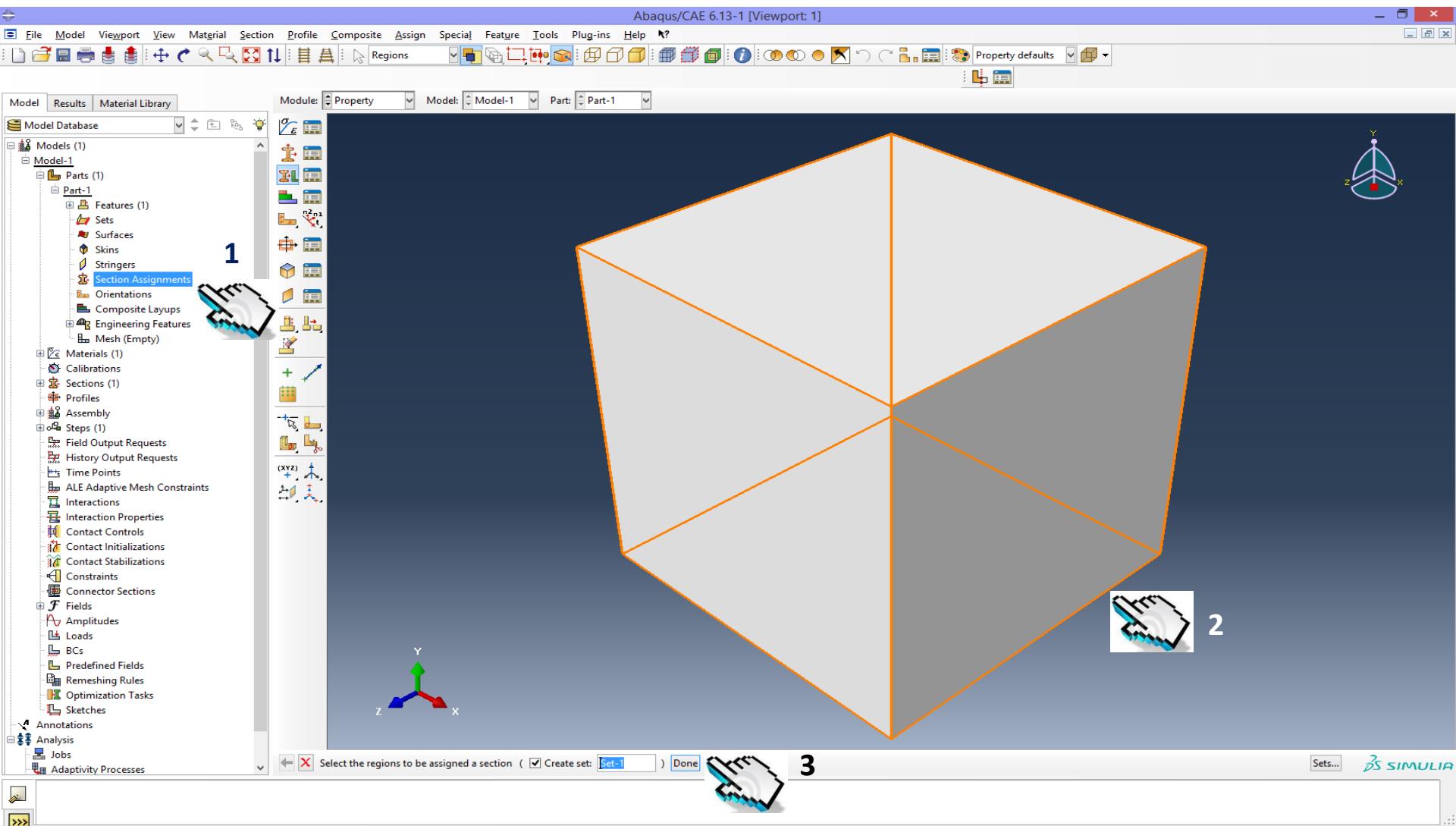
Model - Sections

3. Section : type de matériau



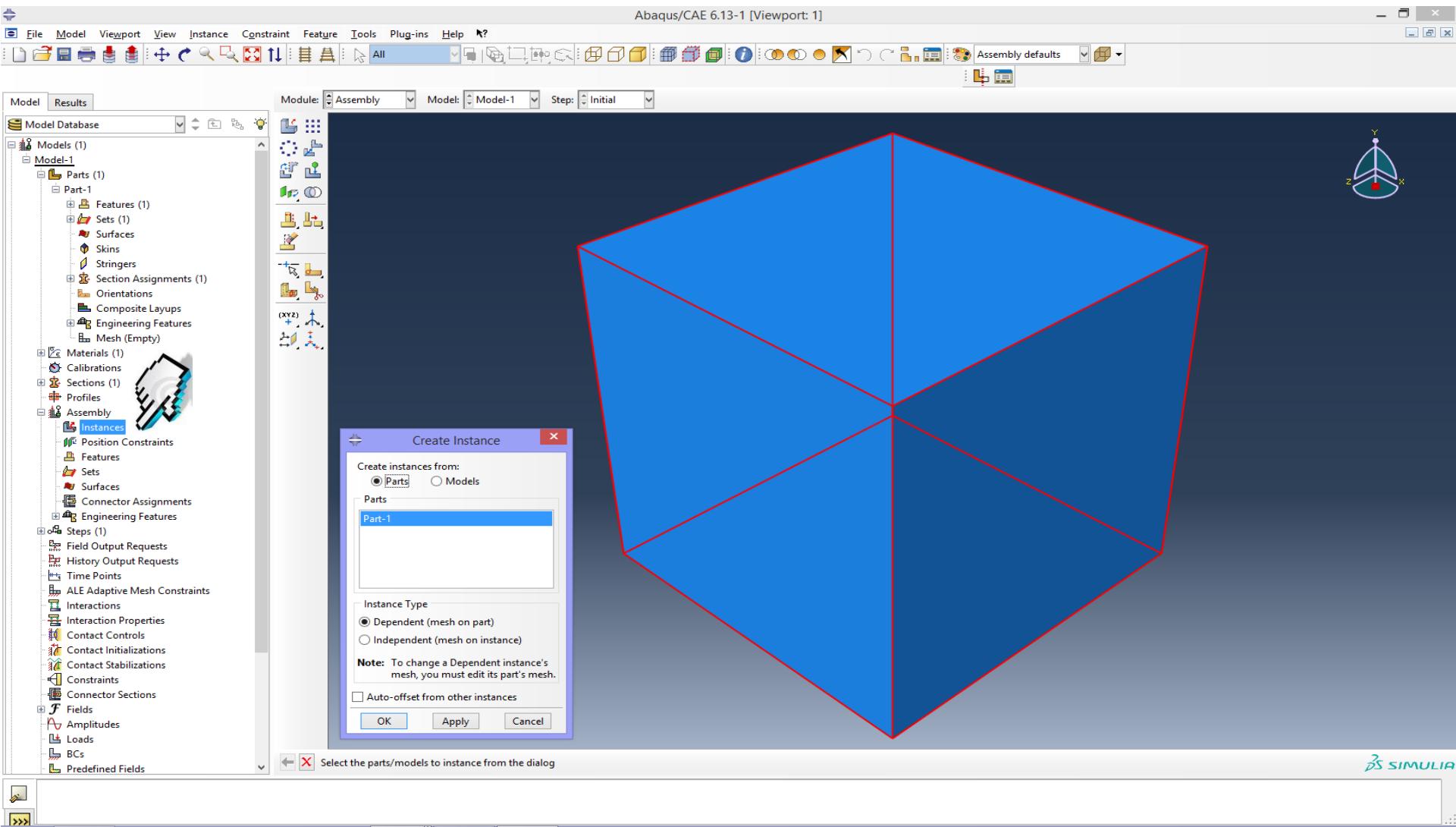
Model - Parts

4. Attribution de la section



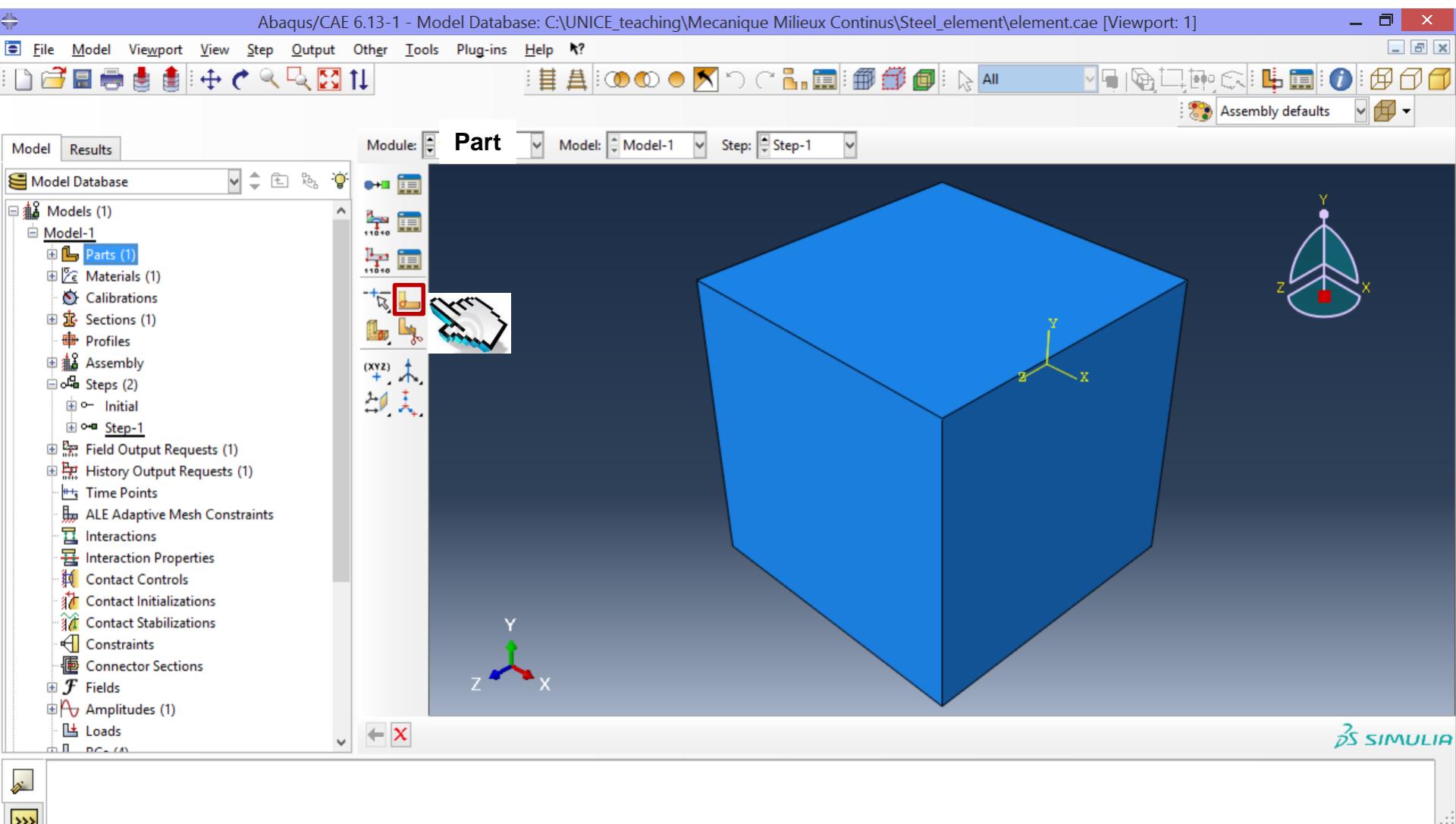
Model - Assembly

5. Assemblage



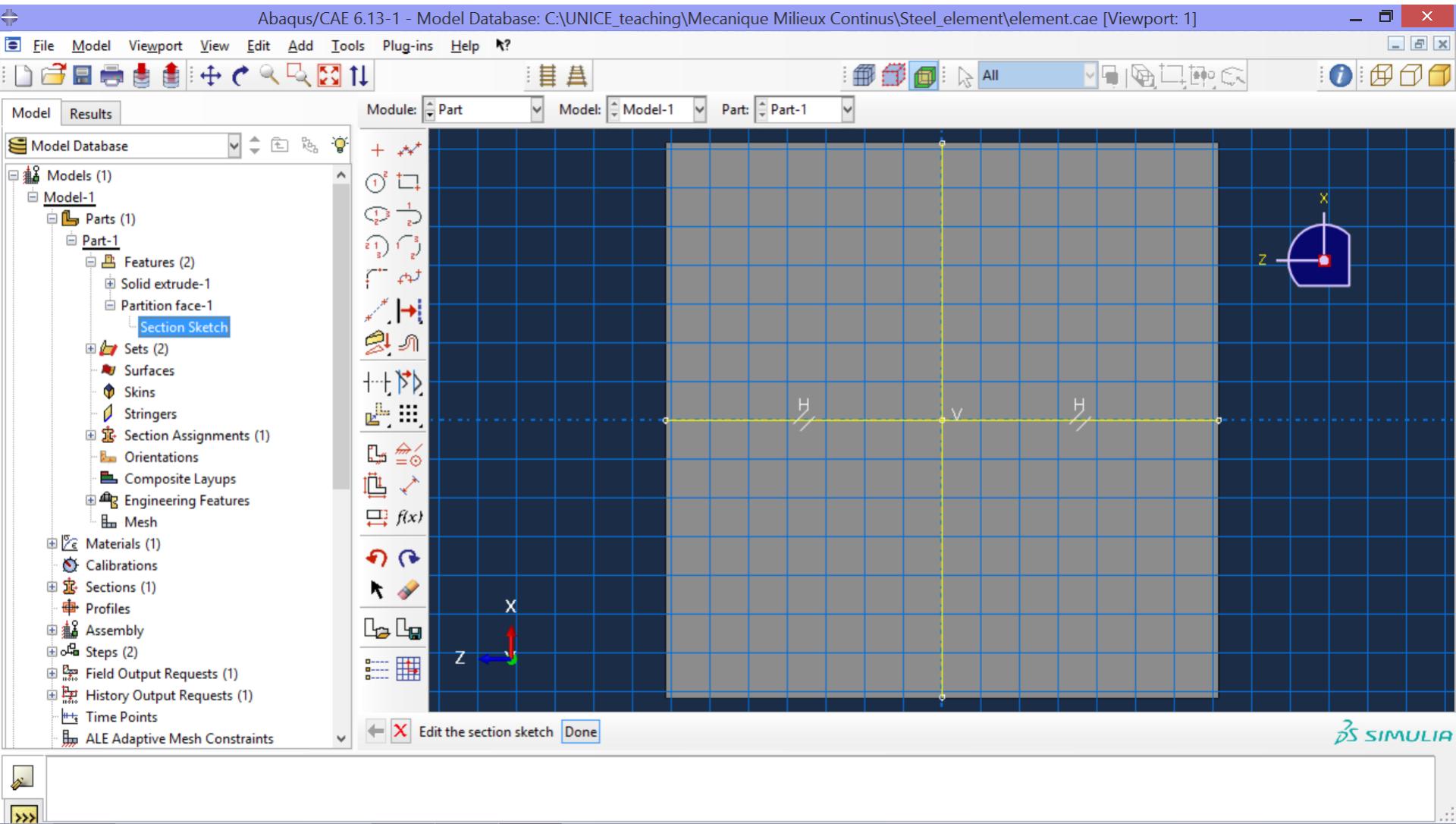
Model - Parts

6. Partitions : dessin des parties où appliquer des conditions



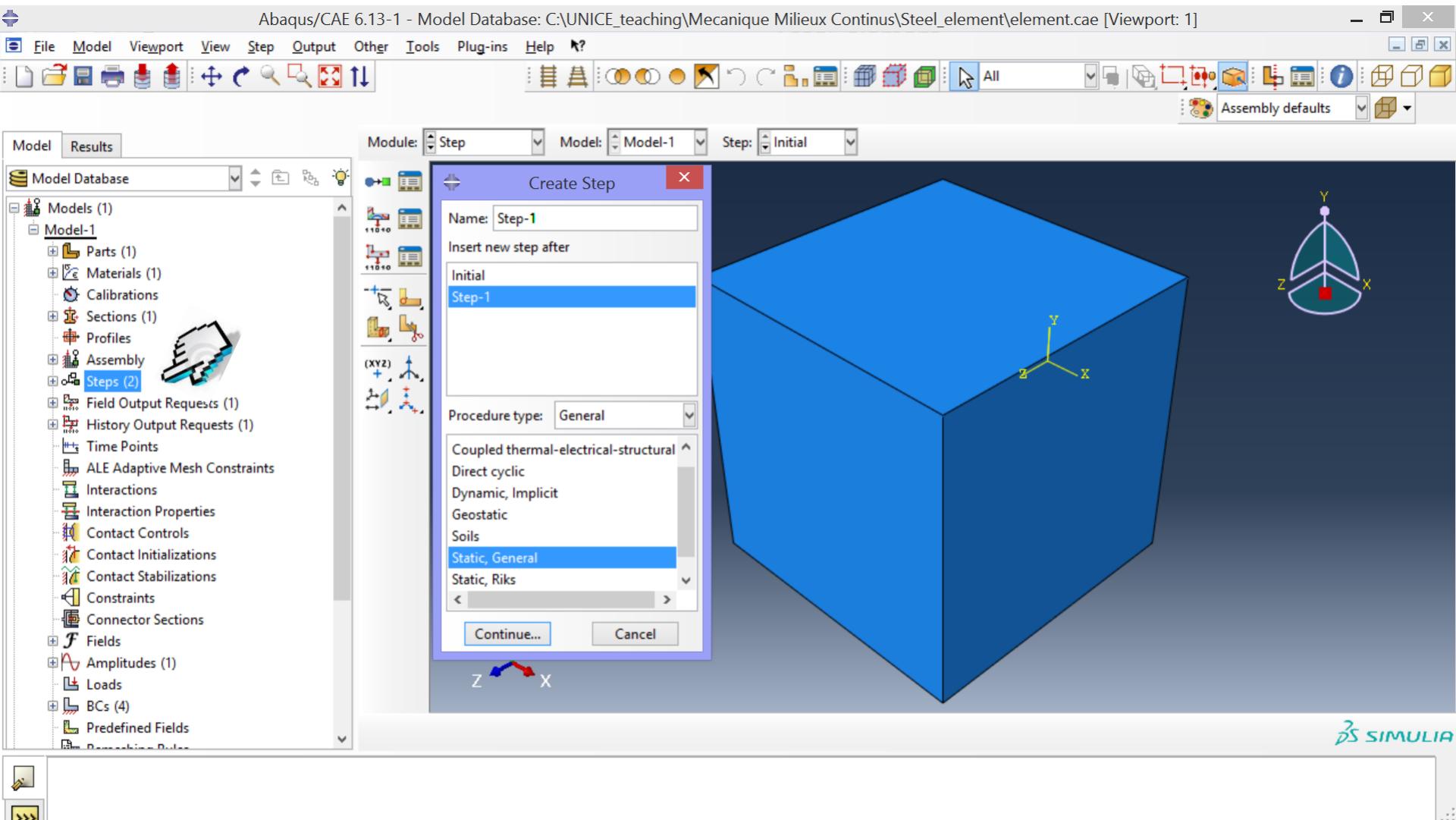
Model - Parts

6. Partitions : dessin des parties où appliquer des conditions



Model - Steps

7. Pas de calcul : Statique, Quasi-statique



Model - Steps

7. Pas de calcul : Statique, Quasi-statique

Abaqus/CAE 6.13-1 - Model Database: C:\UNICE_teaching\Mecanique Milieux Continus\Steel_element\element.cae [Viewport: 1]

File Model Viewport View Step Output Other Tools Plug-ins Help ?

Edit Step Step: Step-1

Name: Step-1
Type: Static, General

Basic Incrementation Other

Type: Automatic Fixed

Maximum number of increments: 60000

Increment size: Initial: 0.01 Minimum: 1E-005 Maximum: 0.01

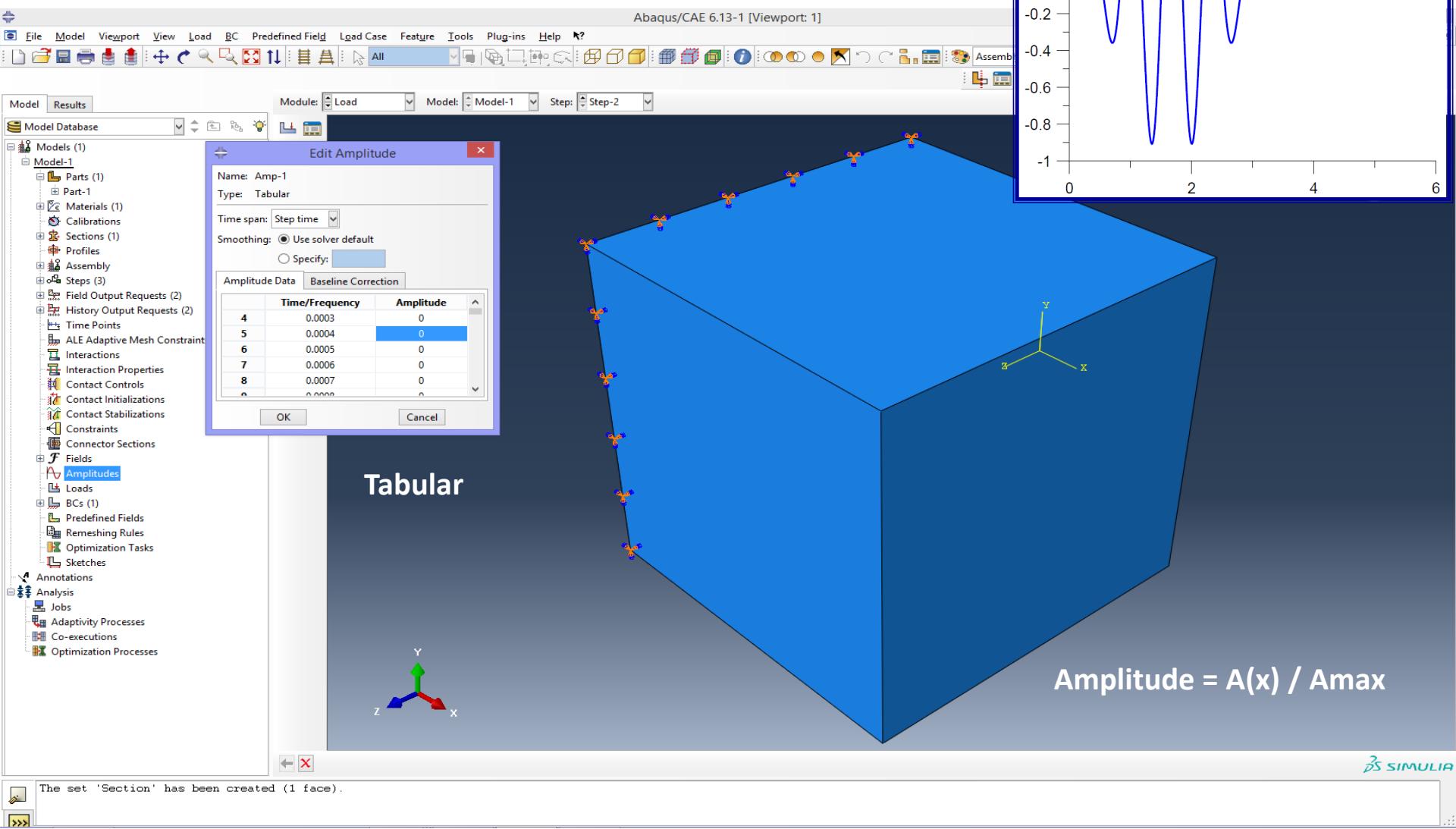
Basic : Time period = durée tmax

Incrementation : pas de temps dt
Max num of increments \geq tmax / dt

DS SIMULIA

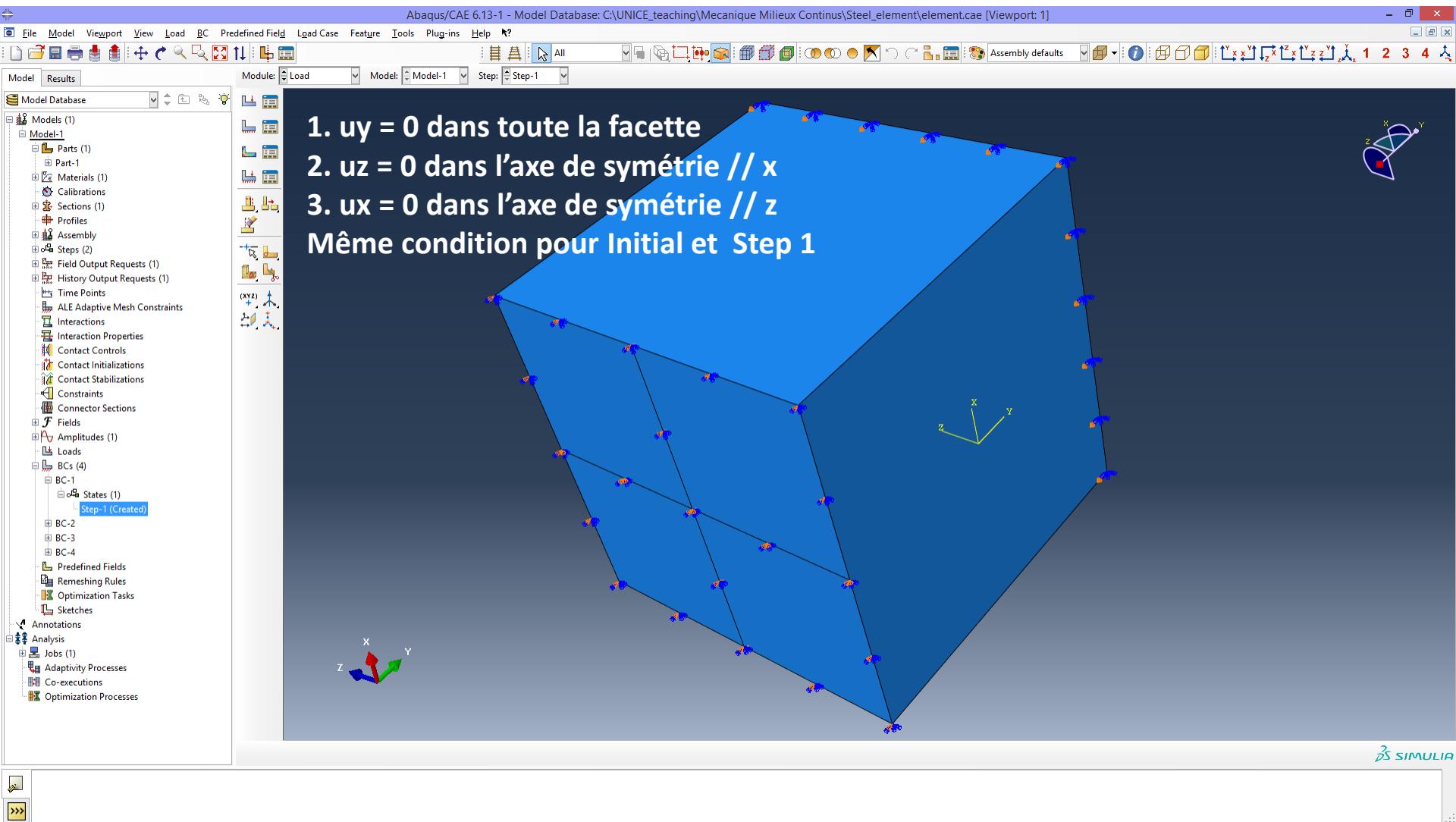
Model - Amplitudes

8. Histoire de charge : amplitude dans le temps



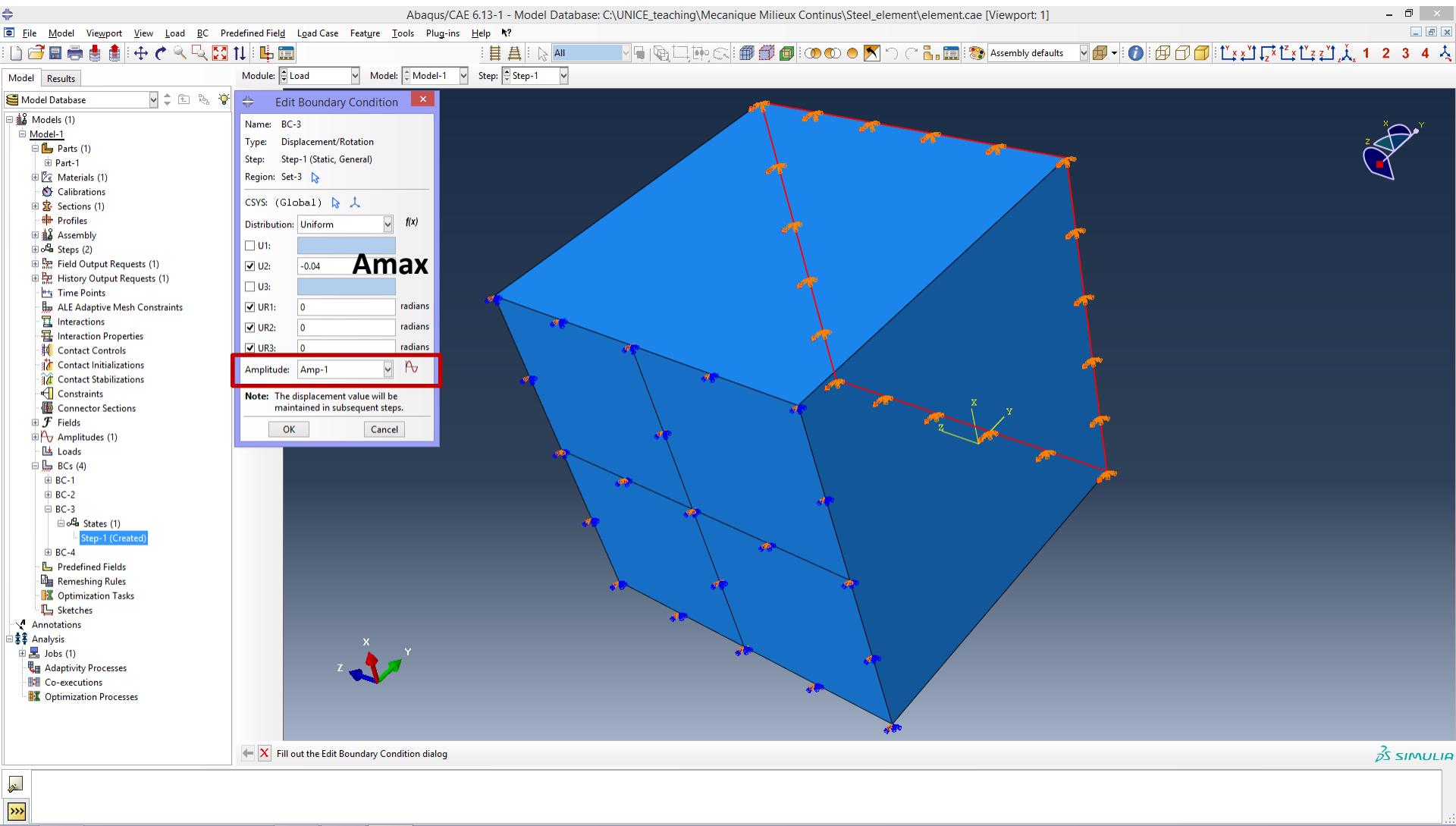
Model - BCs

9. Conditions aux limites : liaisons



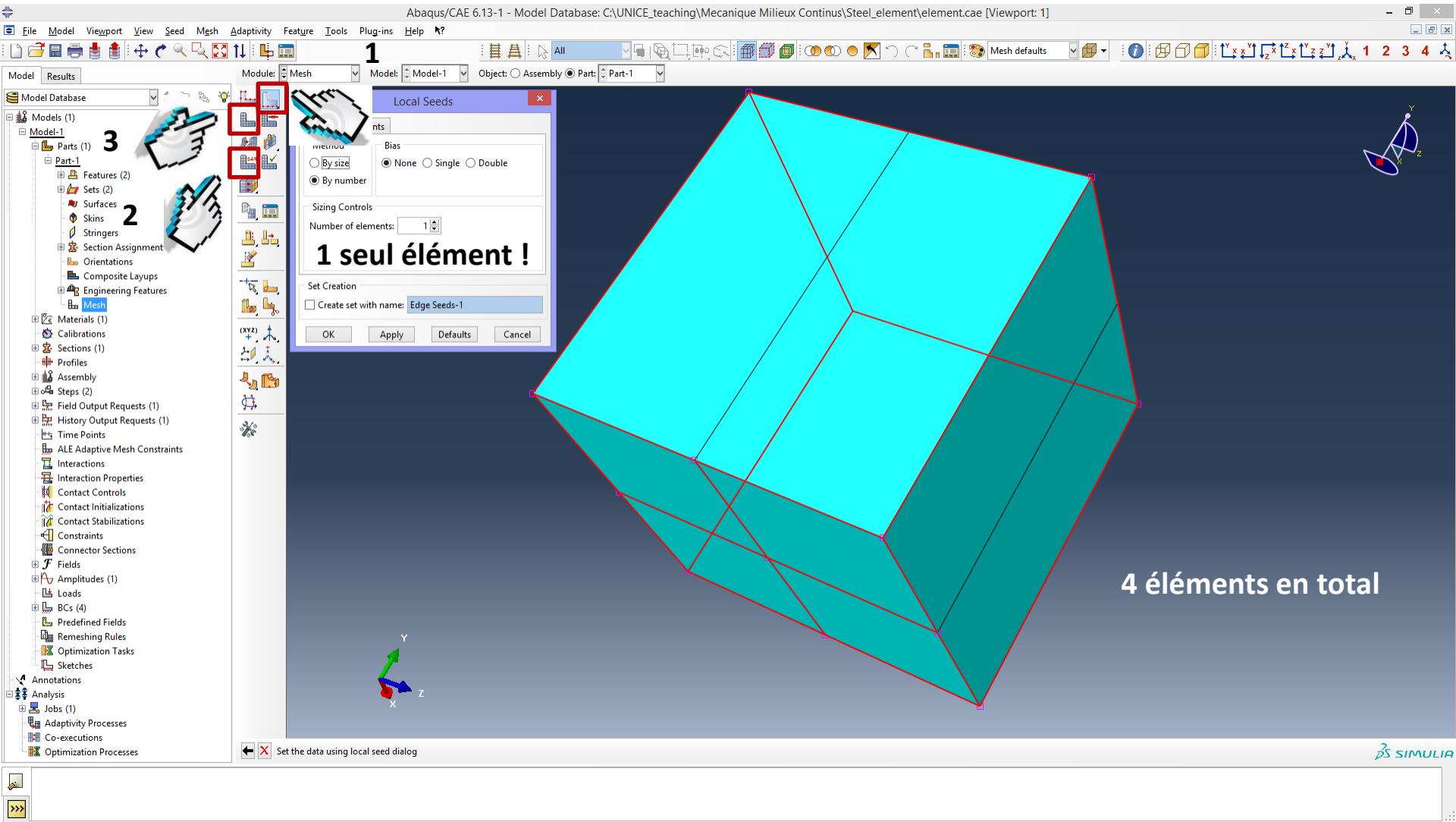
Model - BCs

10. Conditions aux limites : déplacement imposé



Model - Parts

11. Maillage : nombre d'éléments, type d'élément



Model - Field Output Requests

12. Résultats demandés : contraintes, déformations totales, déplacements, réactions

The screenshot shows the Abaqus/CAE 6.13-1 interface. The main window title is "Abaqus/CAE 6.13-1 [Viewport: 1]". The menu bar includes File, Model, Viewport, View, Step, Output, Other, Tools, Plugins, Help, and a question mark icon. The toolbar contains various icons for file operations, selection, and analysis. The left sidebar displays the "Model Database" tree, which includes Model-1, Parts, Materials, Calibrations, Sections, Profiles, Assembly, Steps, Field Output Requests (F-Output-1, F-Output-2), History Output Requests (Time Points, ALE Adaptive Mesh Constraints), Interactions, Interaction Properties, Contact Controls, Contact Initializations, Contact Stabilizations, Constraints, Connector Sections, Fields, Amplitudes, Loads, BCs, Predefined Fields, Remeshing Rules, Optimization Tasks, Sketches, Annotations, Analysis, Jobs, Adaptivity Processes, Co-executions, and Optimization Processes. A cursor is pointing at the "F-Output-2" entry in the "Field Output Requests" section.

The central part of the screen shows the "Edit Field Output Request" dialog box. The "Name" field is set to "F-Output-2", "Step" to "Step-2", and "Procedure" to "Dynamic, Implicit". The "Domain" dropdown is set to "Whole model" (which is highlighted with a red box). The "Frequency" is set to "Every n increments" with "n: 10". The "Timing" is set to "Output at exact times". Under "Output Variables", the "Select from list below" radio button is selected. The list includes "Stresses", "Strains", "Displacement/Velocity/Acceleration" (which is checked), "Forces/Reactions", "Contact", "Energy", "Failure/Fracture", and "Thermal". A note at the bottom states: "Note: Some error indicators are not available when Domain is Whole Model or Implicit". There are also checkboxes for "Output for rebar" and "Output at shell, beam, and layered section points" (with "Use defaults" selected). The "Include local coordinate directions when available" checkbox is checked. At the bottom of the dialog are "OK" and "Cancel" buttons.

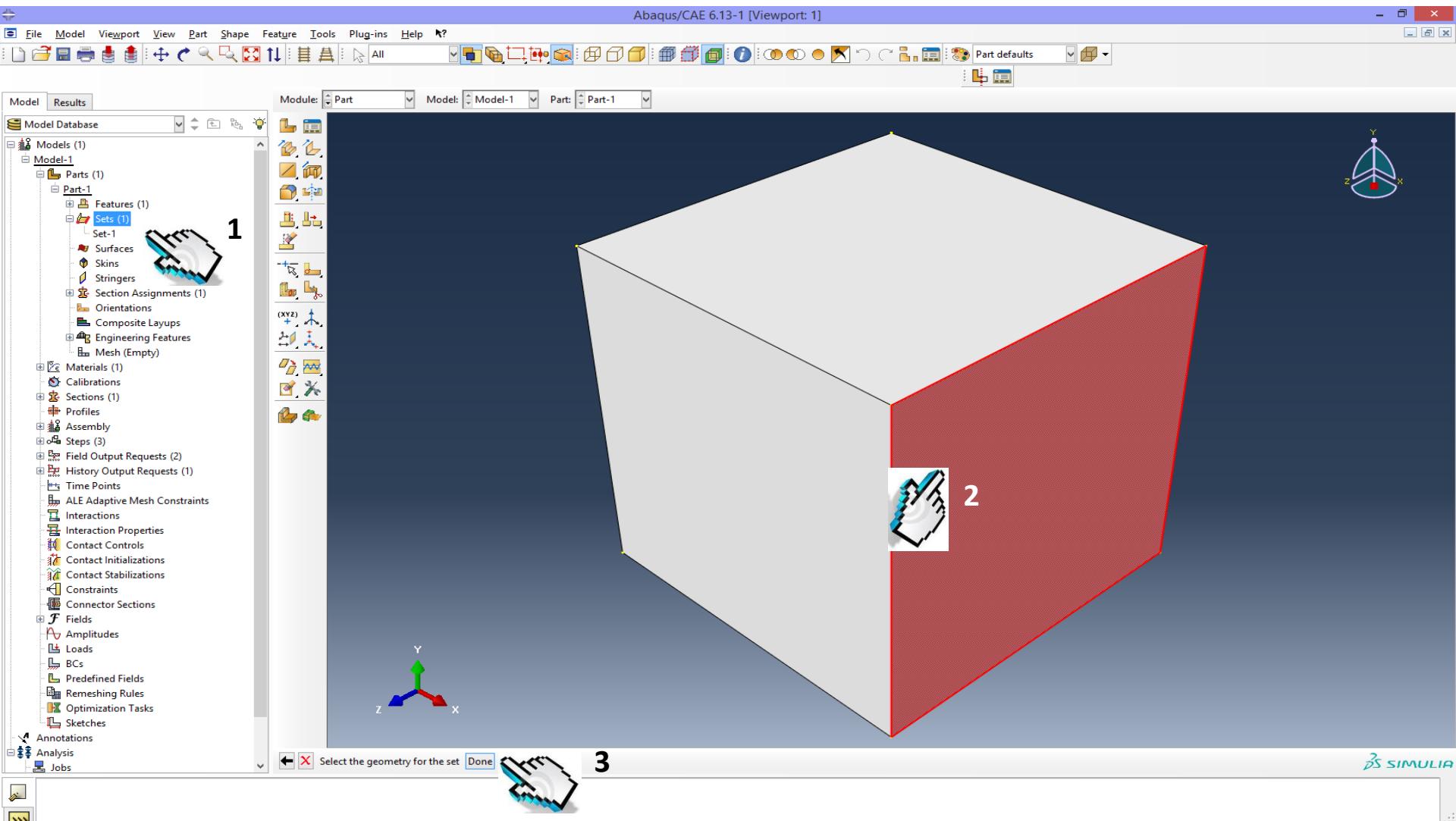
The 3D view on the right shows a blue cube with a coordinate system (X, Y, Z) attached to one of its faces. A small circular icon with a coordinate system is also visible in the top right corner of the 3D view.

Step 1 : contraintes, déformations totales, déplacements

DS SIMULIA

Model - Parts

13. Point où enregistrer les résultats : point, bord, surface



Model - History Output Requests

14. Résultats dans le temps : contrainte, déformation \equiv déplacement

Abaqus/CAE 6.13-1 [Viewport: 1]

File Model Viewport View Step Output Other Tools Plug-ins Help ?

Model Results

Model Database

Models (1)
Model-1
Parts (1)
Part-1
Materials (1)
Calibrations
Sections (1)
Profiles
Assembly
Steps (3)
Field Output Requests (2)
History Output Requests (1)
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches
Annotations

Analysis
Jobs
Adaptivity Processes
Co-executions
Optimization Processes

Module: Step Model: Model-1 Step: Step-2

Edit History Output Request

Name: H-Output-2
Step: Step-2
Procedure: Dynamic, Implicit

Domain: Set : Part-1-1.Section

Frequency: Every n increments n: 1

Timing: Output at exact times

Output Variables
Select from list below: Prespecified defaults All Edit variables
S11,S22,S33,E11,E22,E33,U1,U2,U3

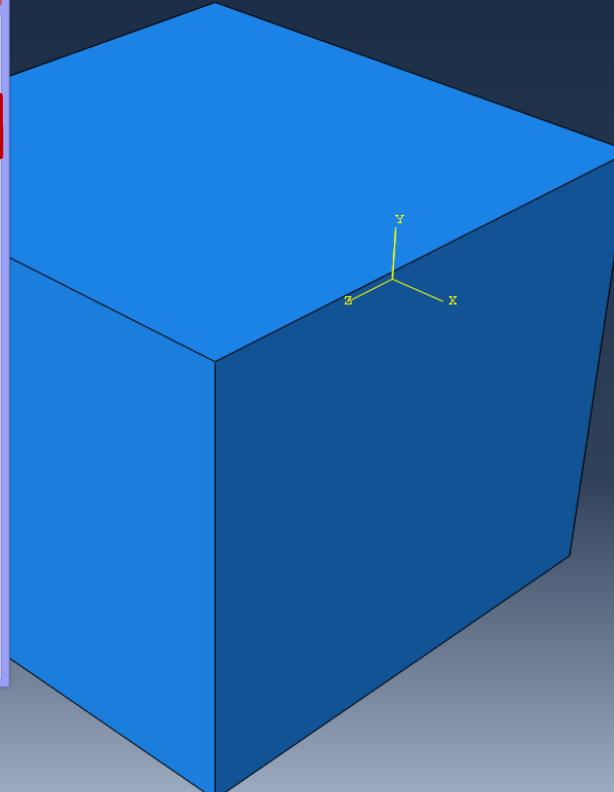
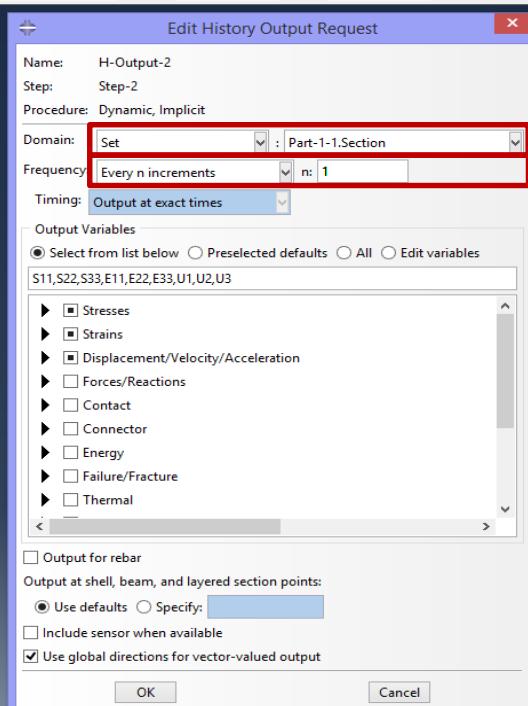
Stresses
 Strains
 Displacement/Velocity/Acceleration
 Forces/Reactions
 Contact
 Connector
 Energy
 Failure/Fracture
 Thermal

Output for rebar

Output at shell, beam, and layered section points:
 Use defaults Specify:
 Include sensor when available
 Use global directions for vector-valued output

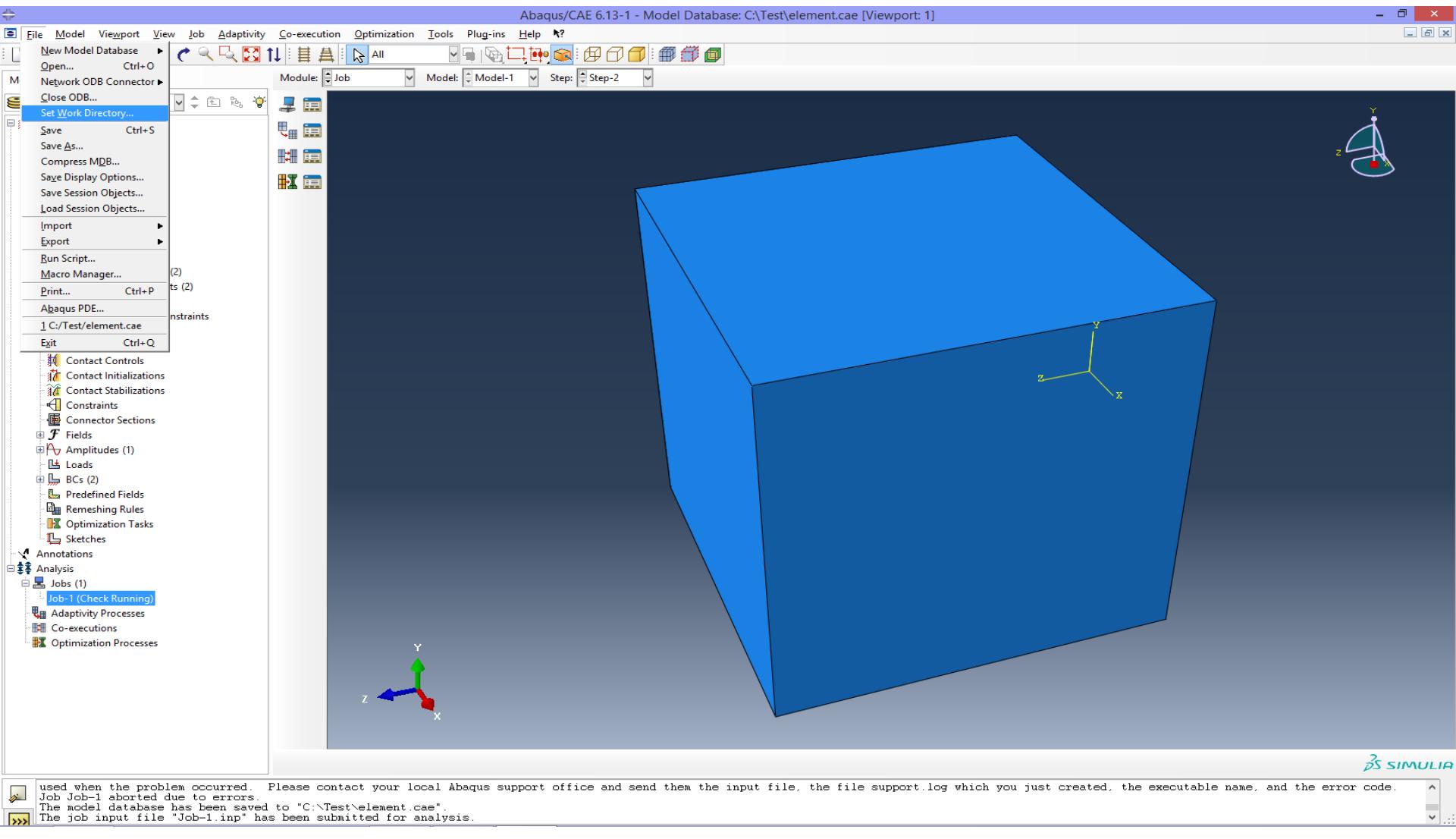
OK Cancel

The set 'Section' has been created (1 face).



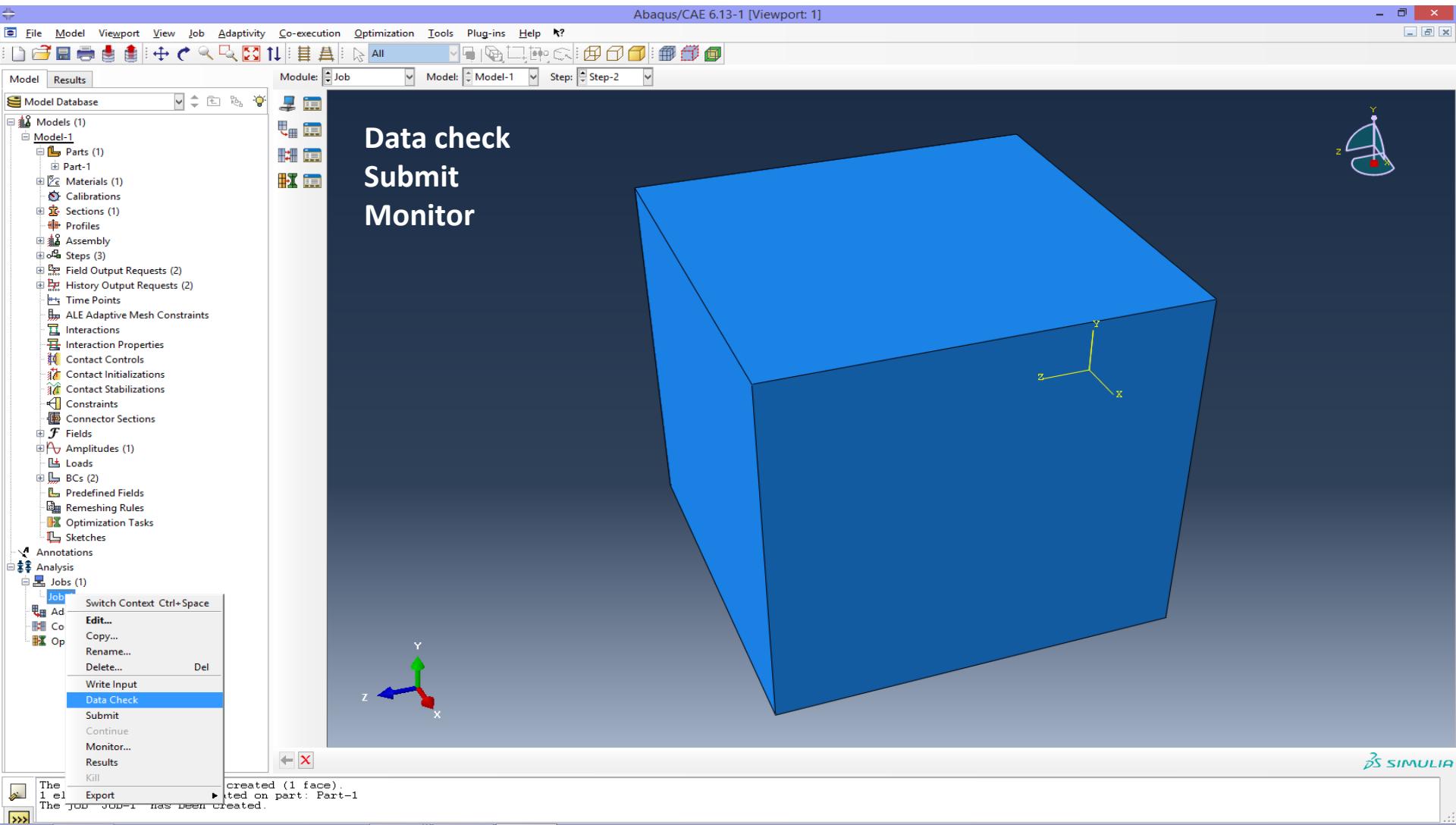
File - Save

15. Sélection du dossier et sauvegarde

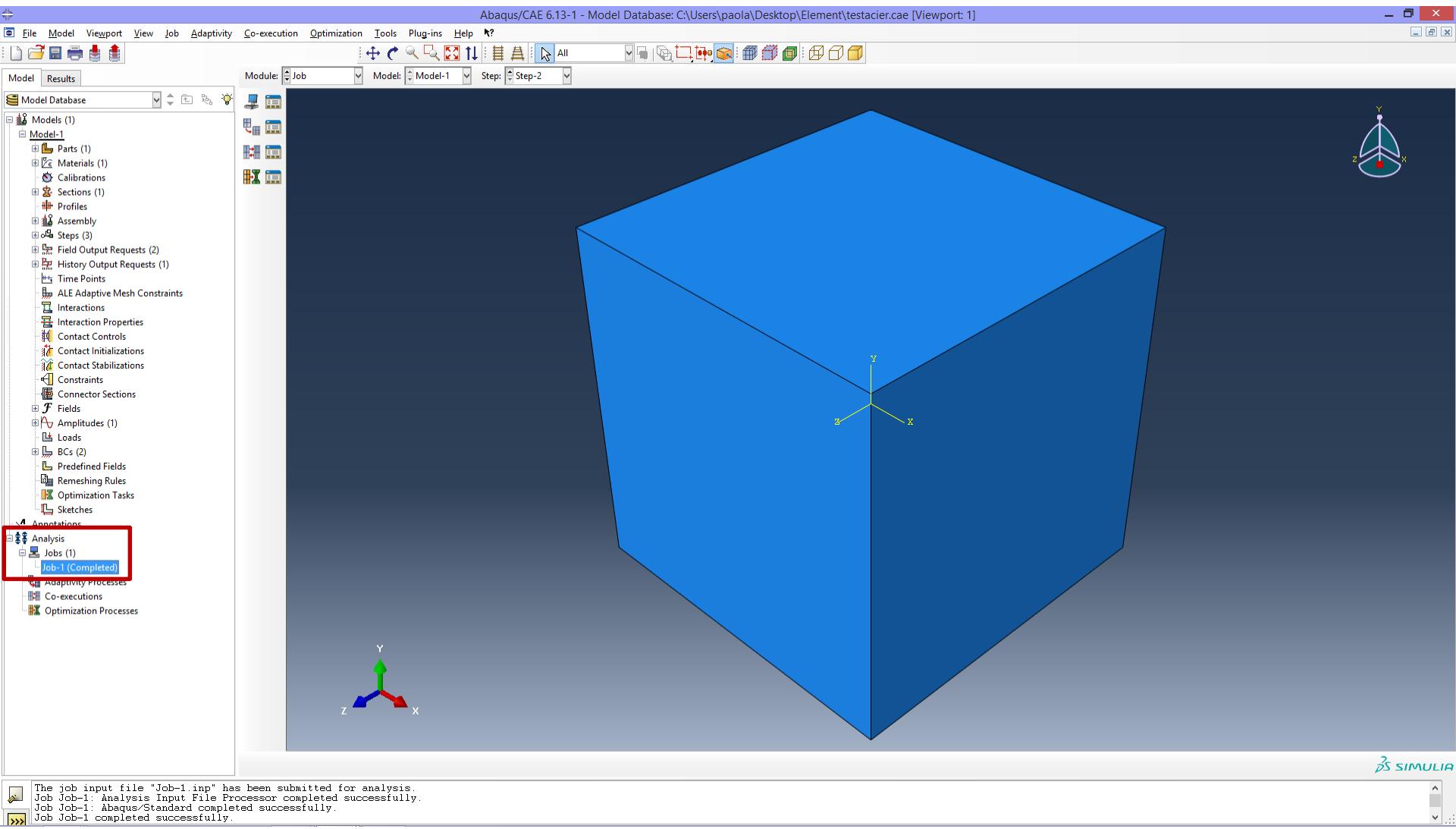


Model - Analysis

16. Création du projet, vérification des données et calcul

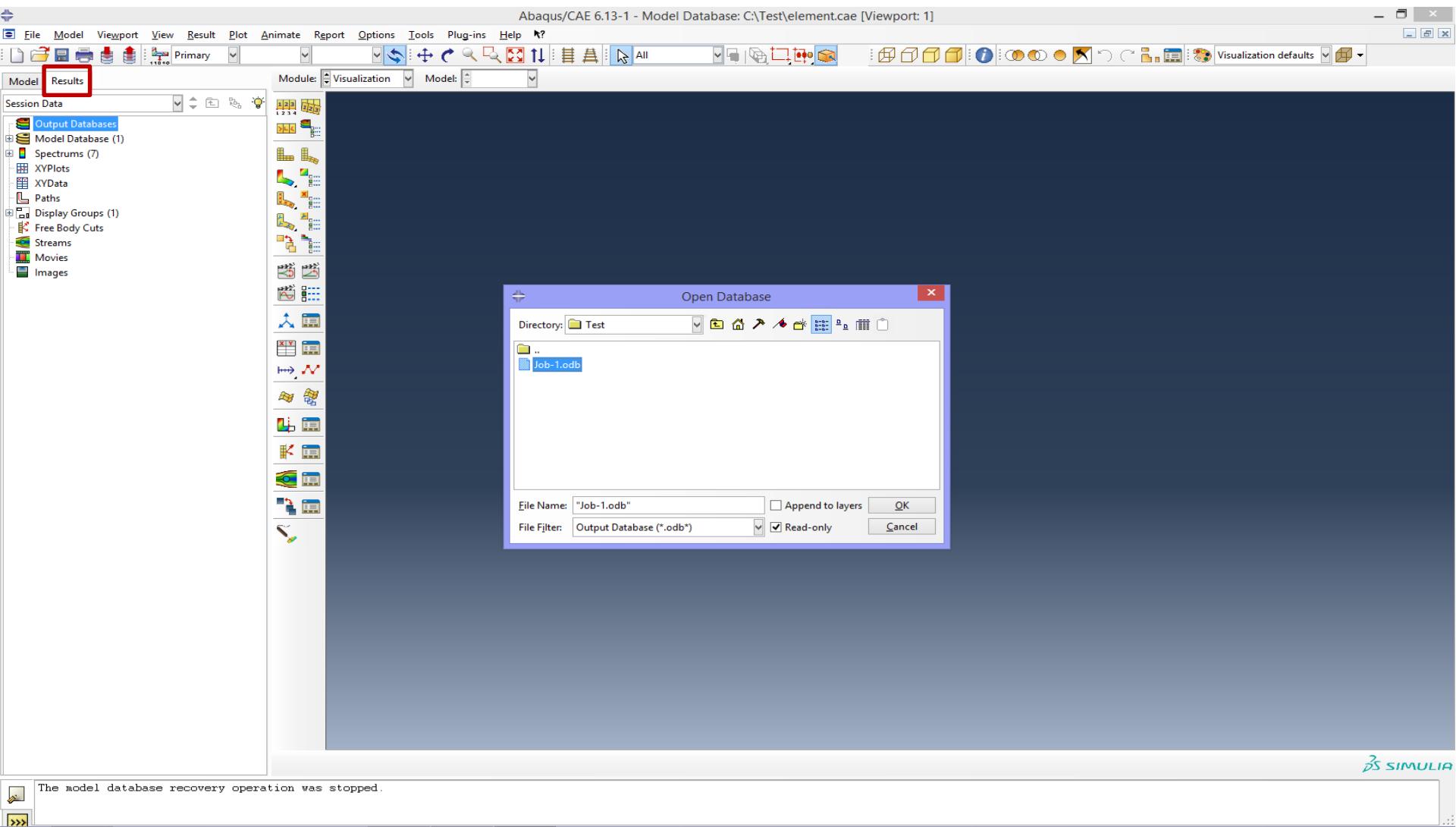


Calcul complet



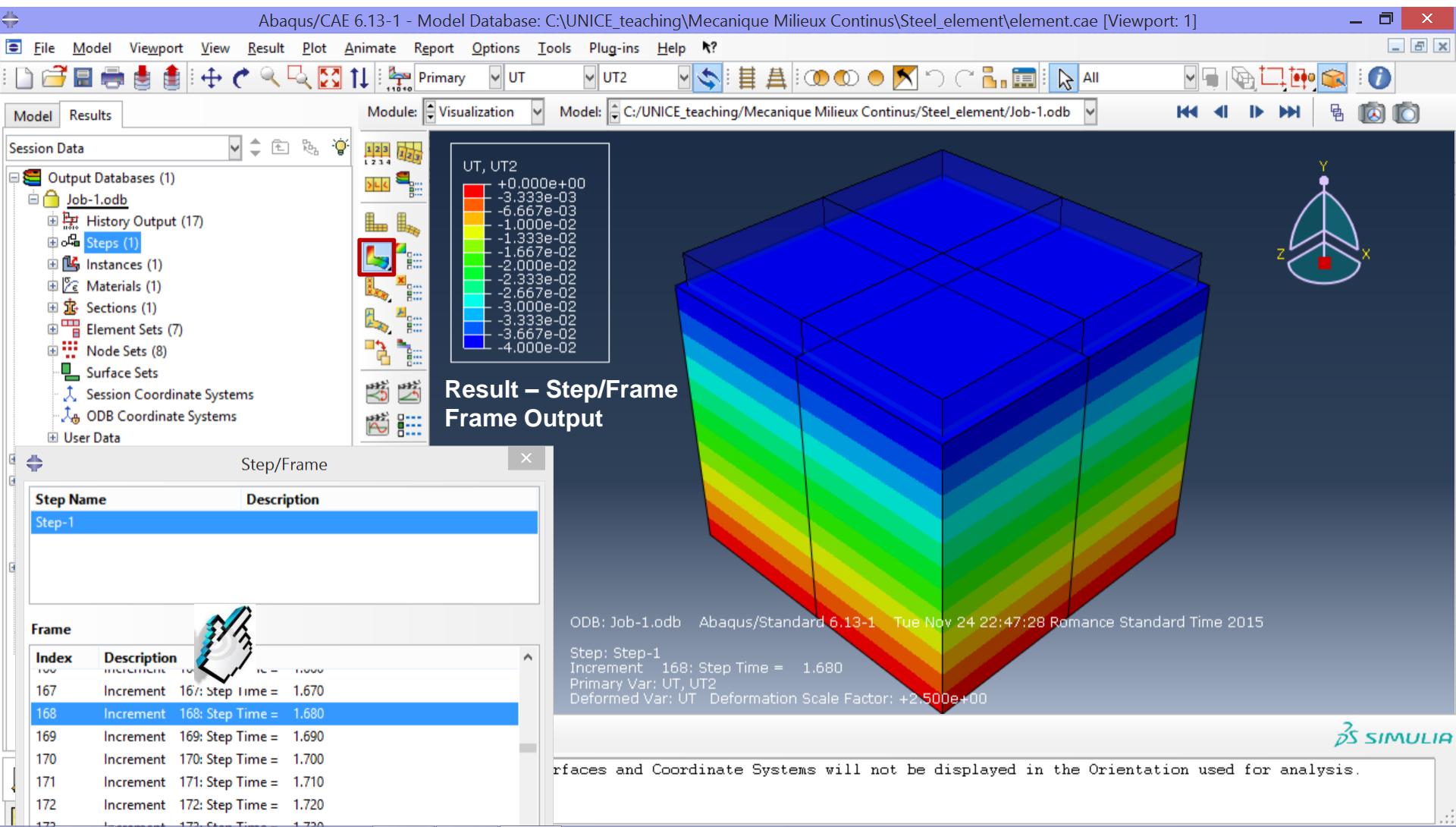
Results

1. Fichier de résultats : *.odb



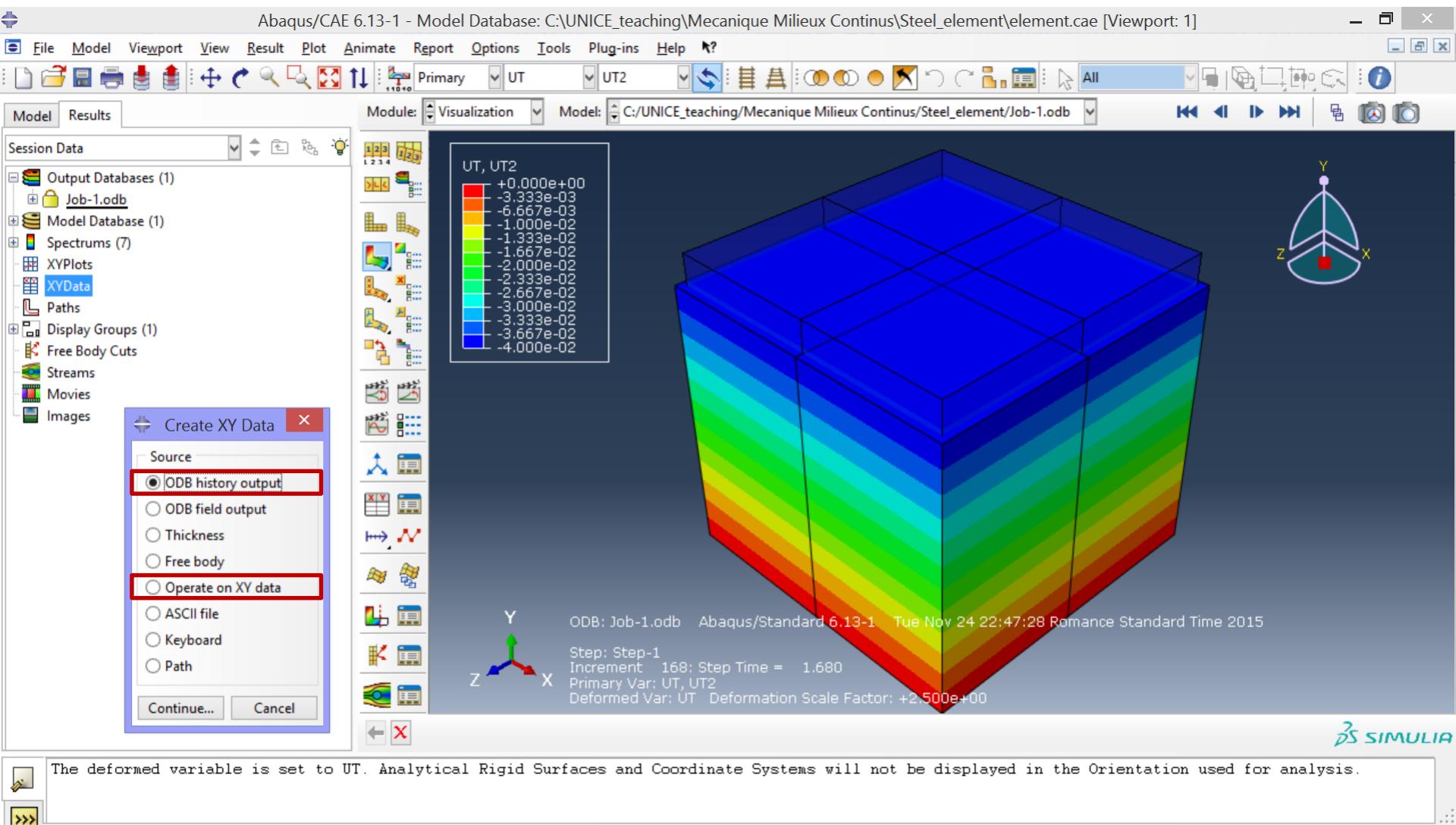
Results

2. Déformée et cartographie : déformation, contrainte



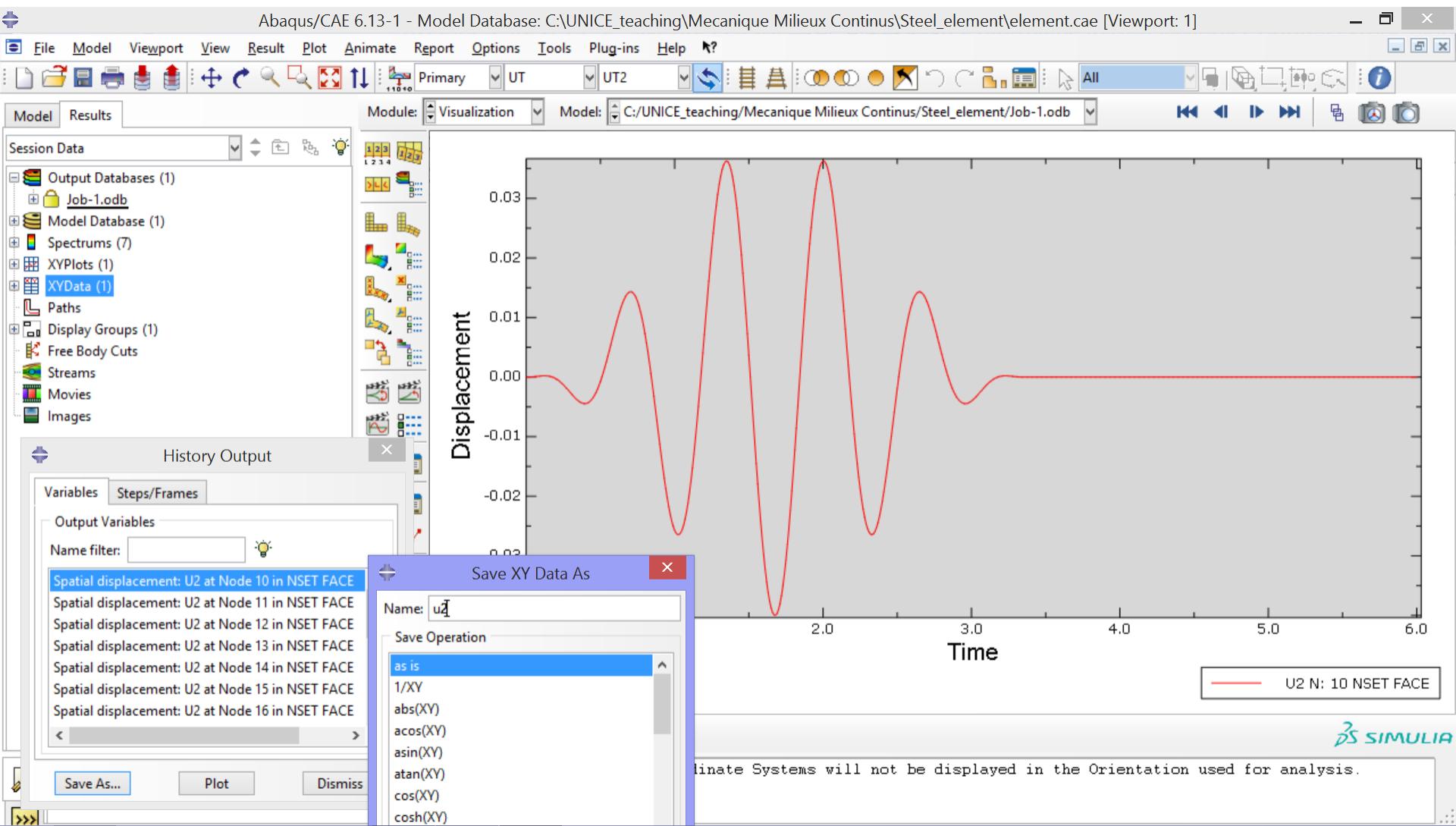
Results - XY Data

3. Courbes : déformation-temps, contrainte-déformation



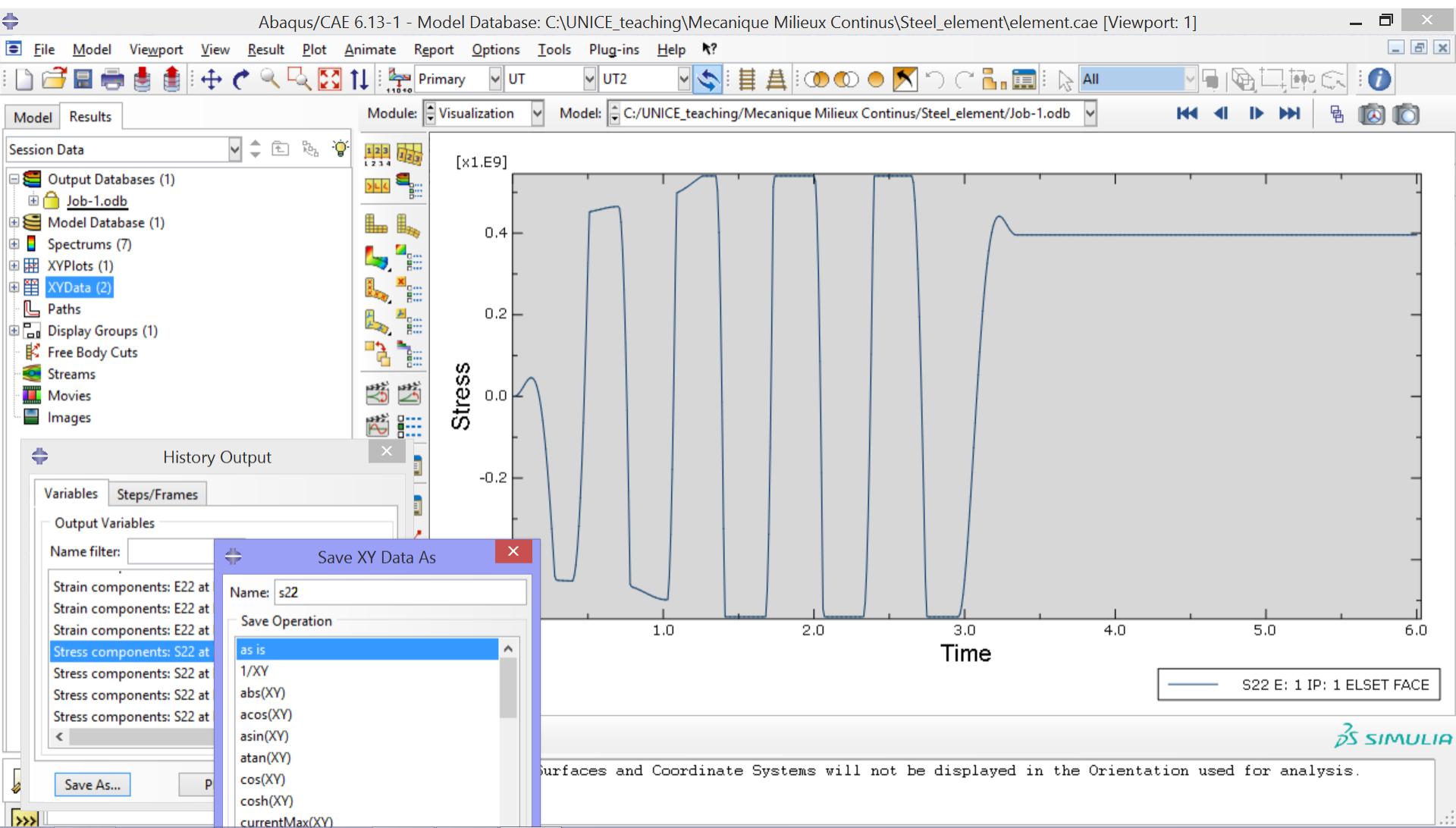
Results - XY Data

3. Courbes : déformation imposée



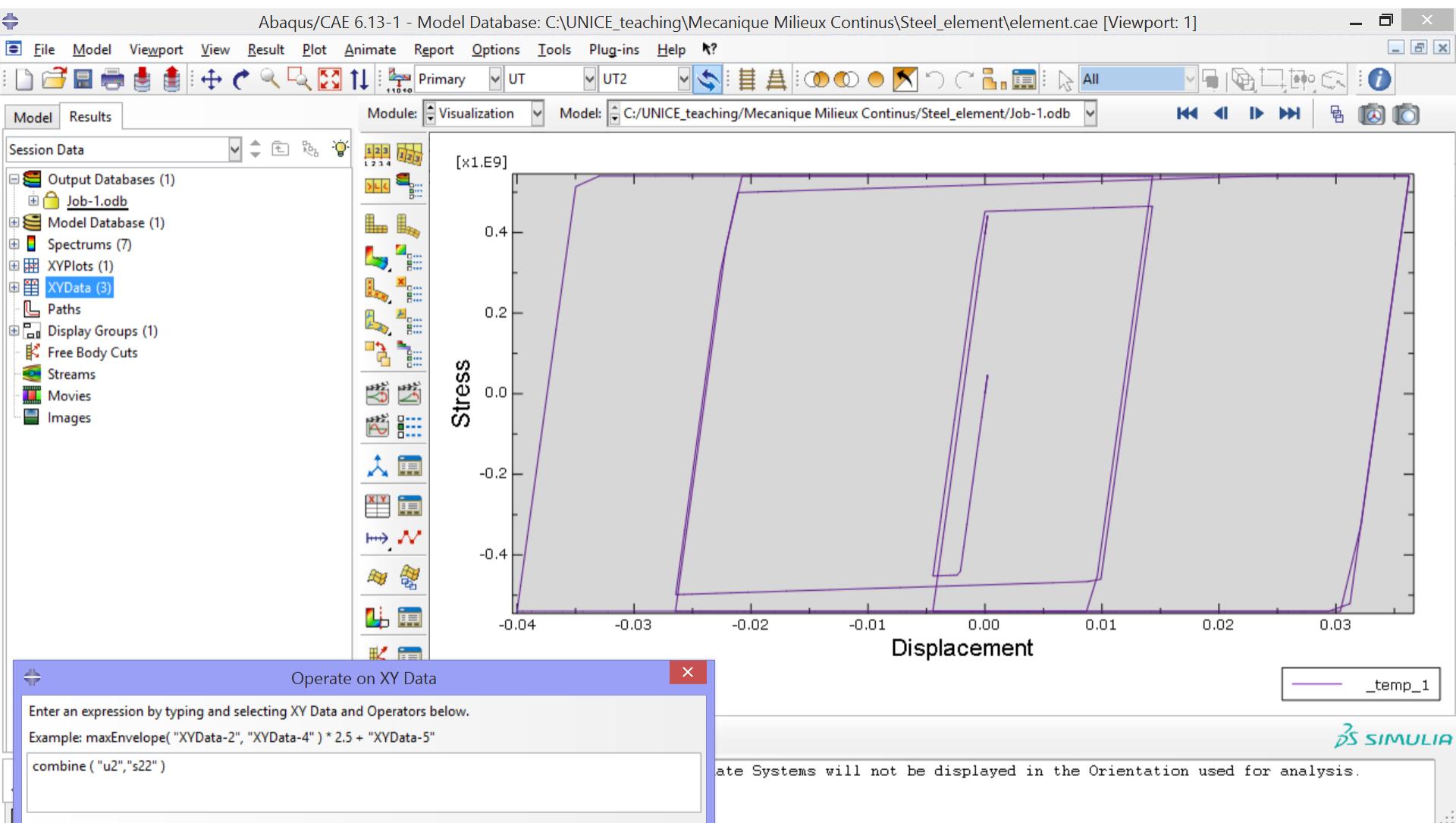
Results - XY Data

3. Courbes : contrainte dans le temps



Results - XY Data

3. Courbes : contrainte-déformation



Fichiers dans le dossier de calcul

Vérifier les données :

- File ***.cae**
- File ***.inp**

Résultats :

- File ***.dat** (Getting Started 7.1.2)
- File ***.odb**

Messages d'erreurs :

- File ***.log**
- File ***.msg**

Loi de comportement 1/2

- **Données** : élément cubique unitaire pour tester les lois de comportement

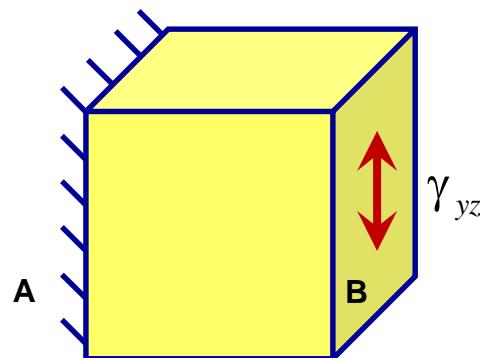
- Géométrie **3D** : $1 \times 1 \times 1$ m
- Matériau : acier (élastique-parfaitement plastique)

Module de Young $E = 210 \cdot 10^9$ N/m², Coefficient de Poisson $\nu = 0.3$, Densité $\rho = 8$ t/m³

Perte d'élasticité : $\varepsilon_y = 0.002$, $f_y = 450$ N/mm²

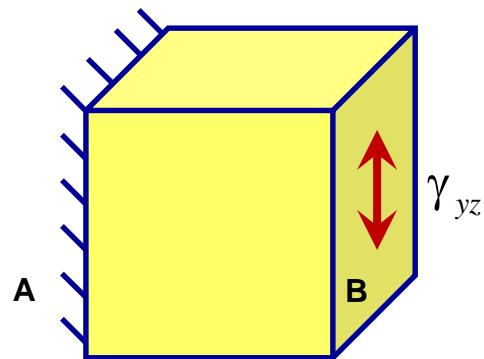
Rupture : $\varepsilon_u = 0.1$, $f_u = 540$ N/mm²

- Conditions aux limites : A) $u_y = 0$, axe de symétrie x : $u_z = 0$, axe de symétrie z : $u_x = 0$
- Contrôle en déformation : B) $u_z \neq 0$ (compression/traction)



Loi de comportement 2/2

- **Données** : élément cubique unitaire pour tester les lois de comportement
 - Condition périodique en direction x : $u_{xA} = u_{xB} \rightarrow du_x = 0 \rightarrow \varepsilon_x = 0$
 - Condition périodique en direction z : $u_{zA} = u_{zB} \rightarrow du_z = 0 \rightarrow \varepsilon_z = 0$



Constraints

Condition périodique : déplacement égal de deux facettes parallèles (tie)

Abaqus/CAE Student Edition 2019 - Model Database: C:\UNICE_teaching\Mecanique Milieux Continus\TD\TD4\Ex1 Steel\shear_four_elements_2019\cube.cae [Viewport: 1]

File Model Viewport View Interaction Constraint Connector Special Feature Tools Plug-ins Help ?

Assembly defaults Module: Interaction Model: Model-1 Step: Initial

Model Results

Model Database

Models (1)
Model-1
Parts (1)
Materials (1)
Calibrations
Sections (1)
Profiles
Assembly
Steps (2)
Field Output Requests (1)
History Output Requests (1)
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints (2)
Constraint-1
Constraint-2
Connector Sections
Fields
Amplitudes

Edit Constraint

Name: Constraint-1
Type: Tie
Master surface: m_Set-15
Slave surface: s_Set-15
Discretization method: Analysis default
 Exclude shell element thickness
Position Tolerance
 Use computed default
 Specify distance: 1.1
Note: Nodes on the slave surface that are considered to be outside the position tolerance will NOT be tied.
 Adjust slave surface initial position
 Tie rotational DOFs if applicable
Constraint Ratio
 Use analysis default
 Specify value
OK Cancel

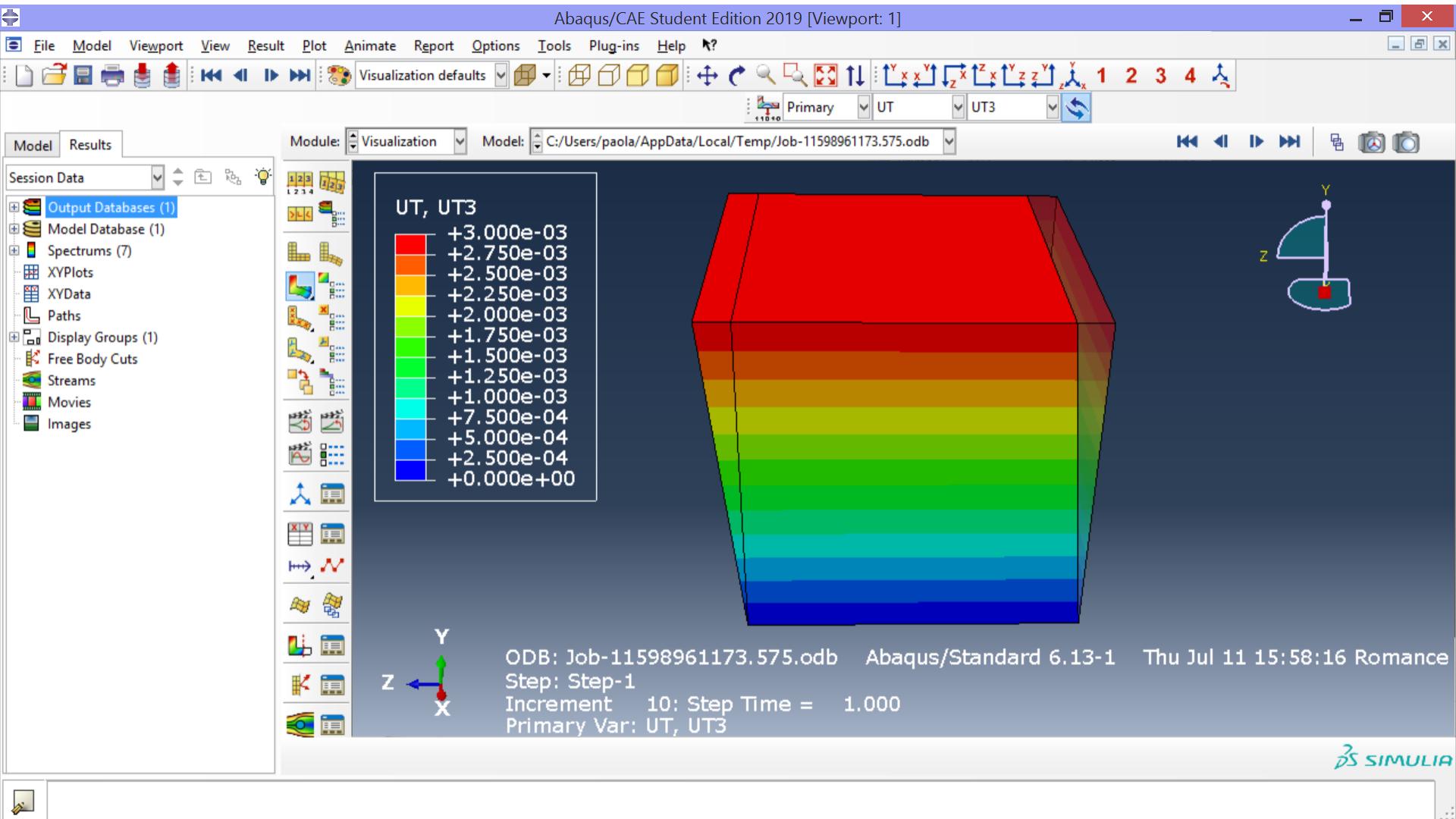
3D Viewport showing a cube model. The front face is blue and the right face is red. Nodes on the front face are highlighted with red circles, and nodes on the right face are highlighted with yellow circles. A callout shows a node on the right face with a red square indicating it is the slave surface. A callout shows a node on the front face with a red circle indicating it is the master surface.

Constraint → Tie
Master surface : node region
Slave surface : node region

DS SIMULIA

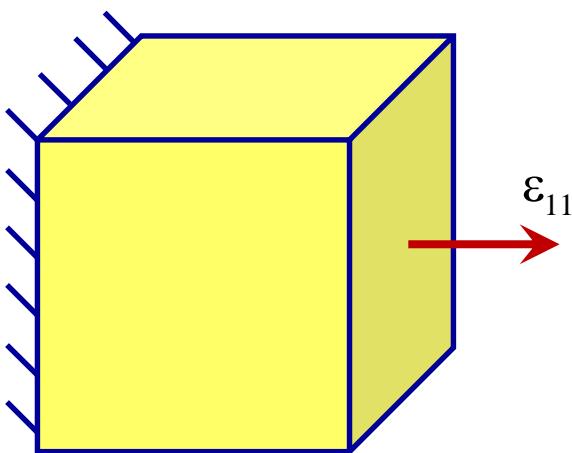
Results

Déformée et cartographie : déformation, contrainte



Loi de comportement 1/2

- **Données** : élément cubique unitaire pour tester les lois de comportement
 - Géométrie **3D** : $1 \times 1 \times 1$ m
 - Matériau : 1) béton, $E = 31220$ N/mm², $\nu = 0.2$, $\rho = 2500$ kg/m³
 - Loi de comportement : élastique-plastique (concrete damage plasticity)
 - Conditions aux limites : A) $u_x = 0$, axe de symétrie y : $u_z = 0$, axe de symétrie z : $u_y = 0$
 - Contrôle en déformation : B) $u_x \neq 0$ (compression/traction)

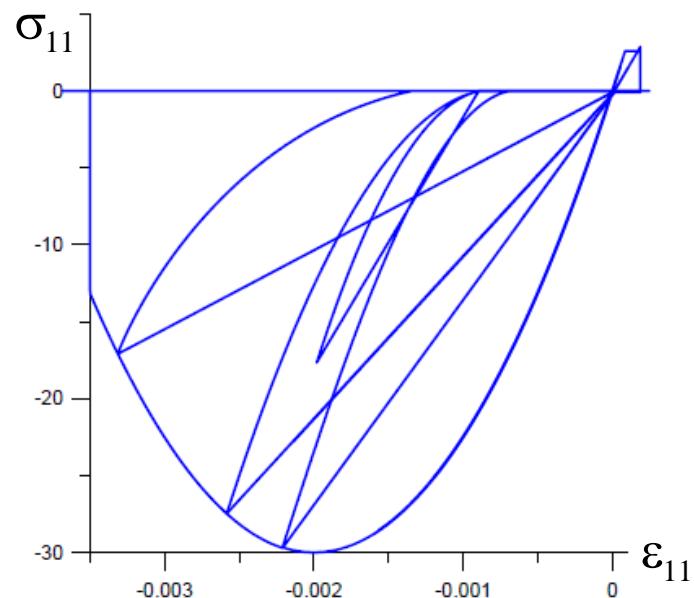
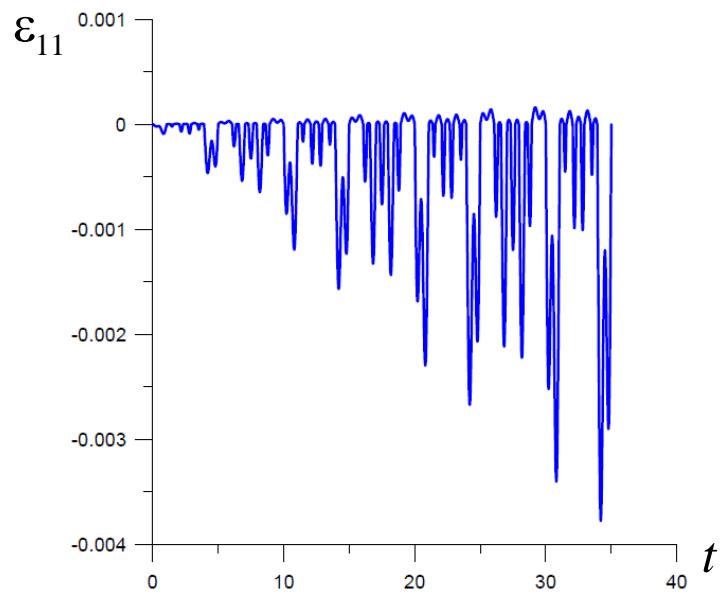
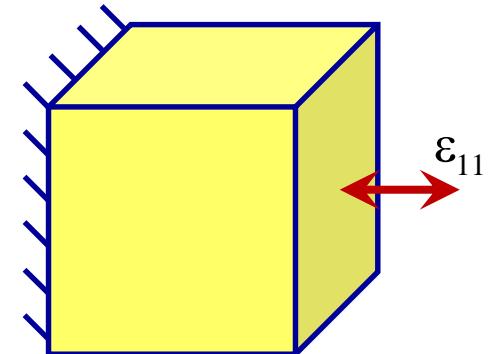


Loi de comportement 2/2

- **Résultats :**

- Courbe déformation-contrainte :

pour le cube unitaire déplacement \equiv contrainte



Results - XY Data

3. Courbes : contrainte-déformation

