

Mécanique des solides

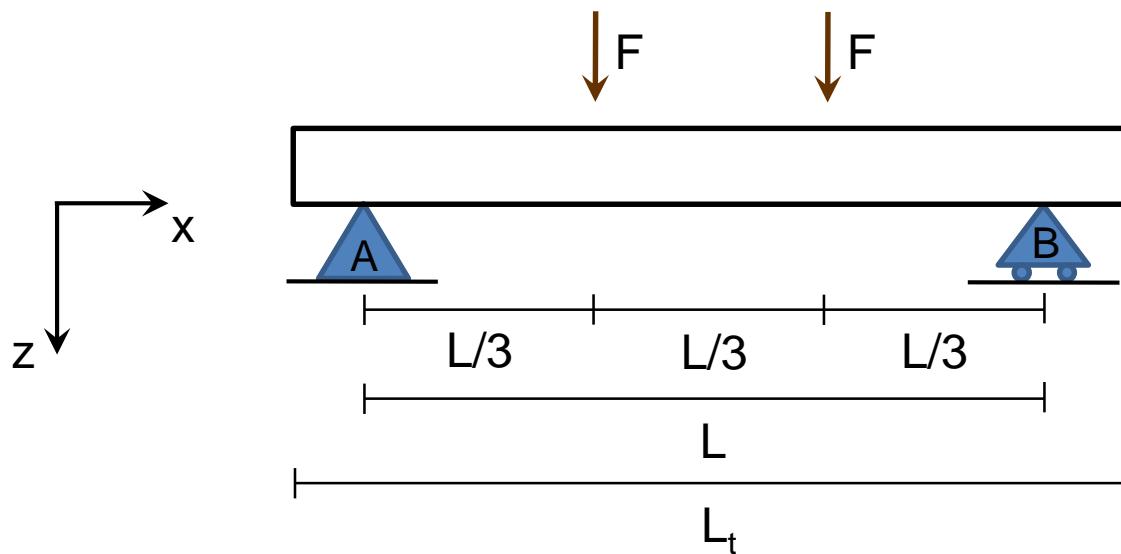
5. Poutre 3D en béton armé

Reine Fares reine.fares@cea.fr

Modélisation en 3D d'une poutre en béton armé

- **Données :**

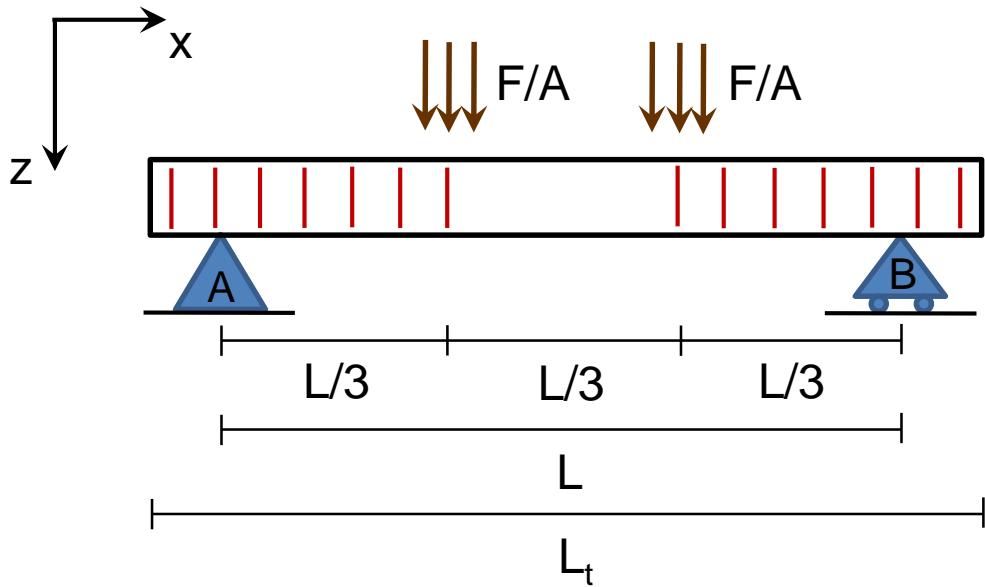
- Géométrie **3D** : béton et barres d'acier. $L_t = 3.5 \text{ m}$, $L = 3.3 \text{ m}$
- Matériau : 1) acier élasto-plastique 2) béton élastique linéaire
- Conditions aux limites : A) $u_z = 0$, $u_x = u_y = 0$, B) $u_z = 0$
- Contrôle en charges : Pression



Modélisation en 3D d'une poutre en béton armé

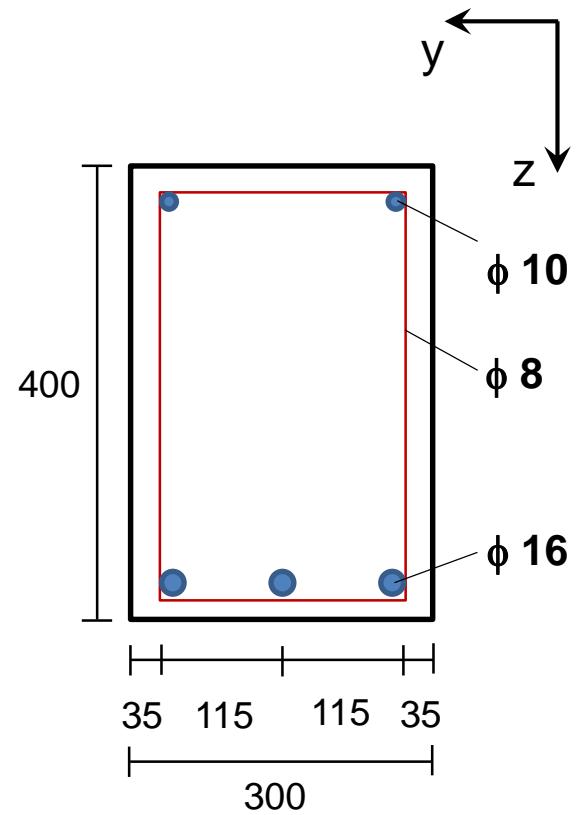
■ Données :

- Contrôle en charges : Pression croissante dans une surface de $300 \times 100 \text{ cm}^2$
- Barres en acier : espacées de 200 mm



Densité du béton : 2400 kg/m^3

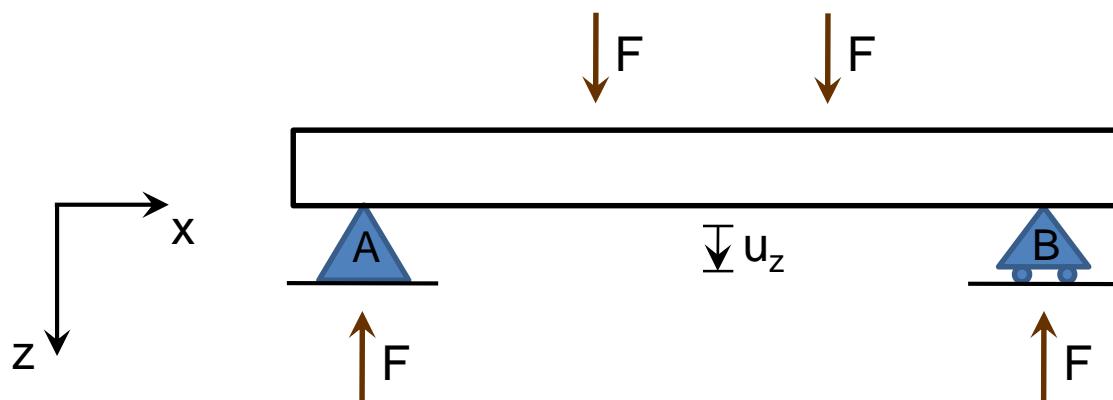
Densité du béton armé : 2500 kg/m^3



Modélisation en 3D d'une poutre en béton armé

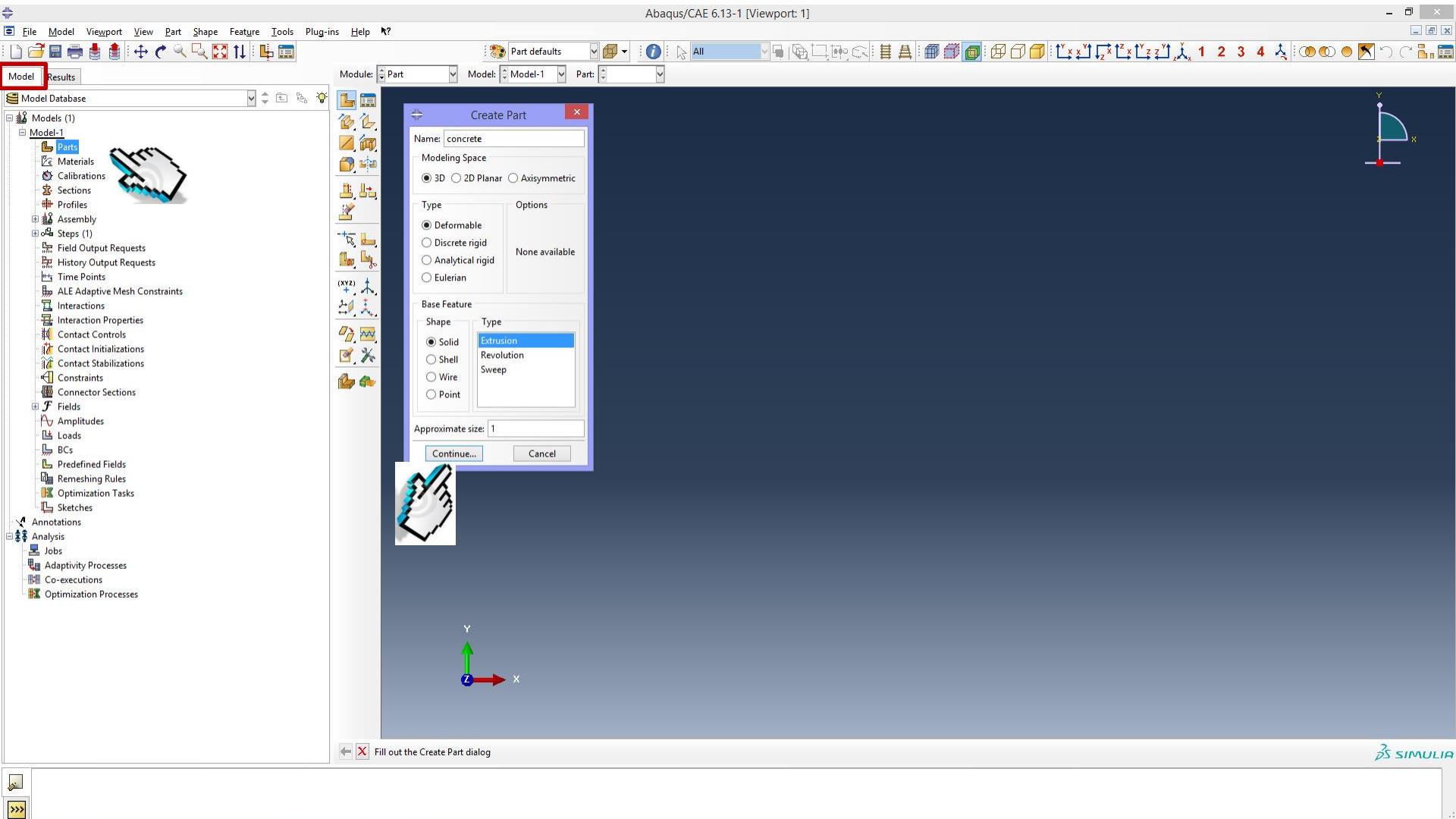
- **Résultats :**

- Flèche, Contraintes, Réactions
- Comportement des barres d'acier



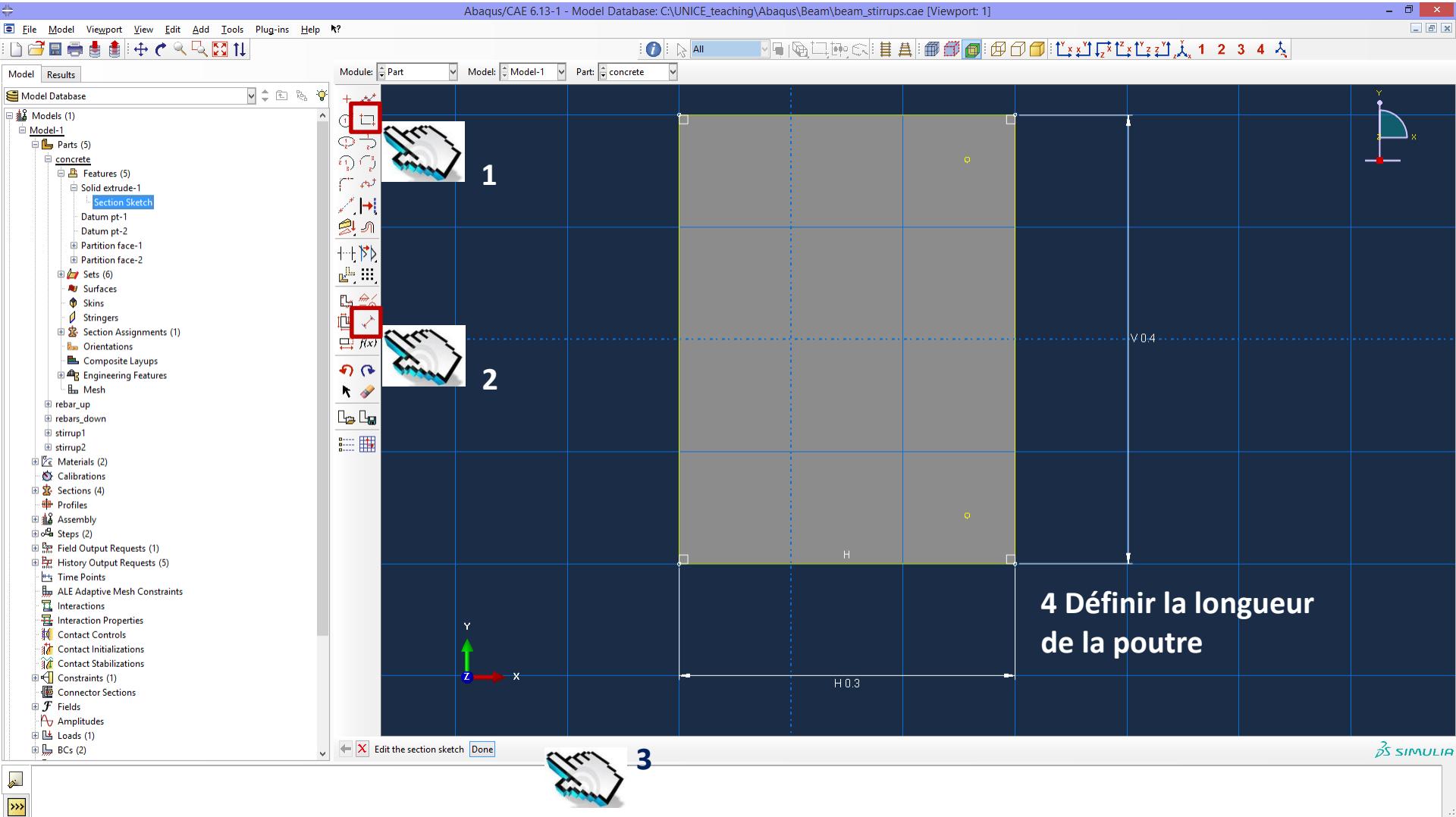
Model - Parts

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



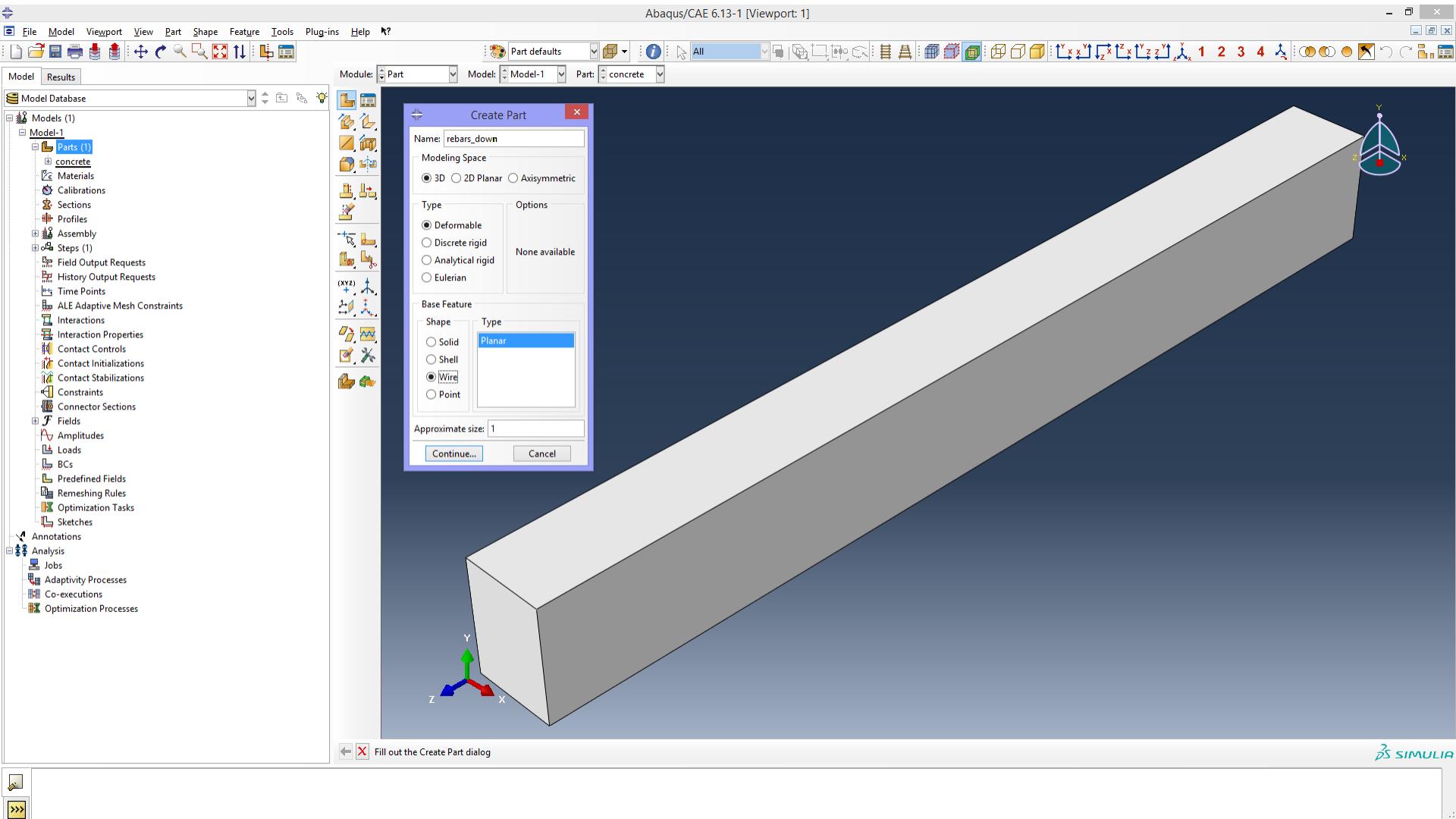
Model - Parts

1. Géométrie : section



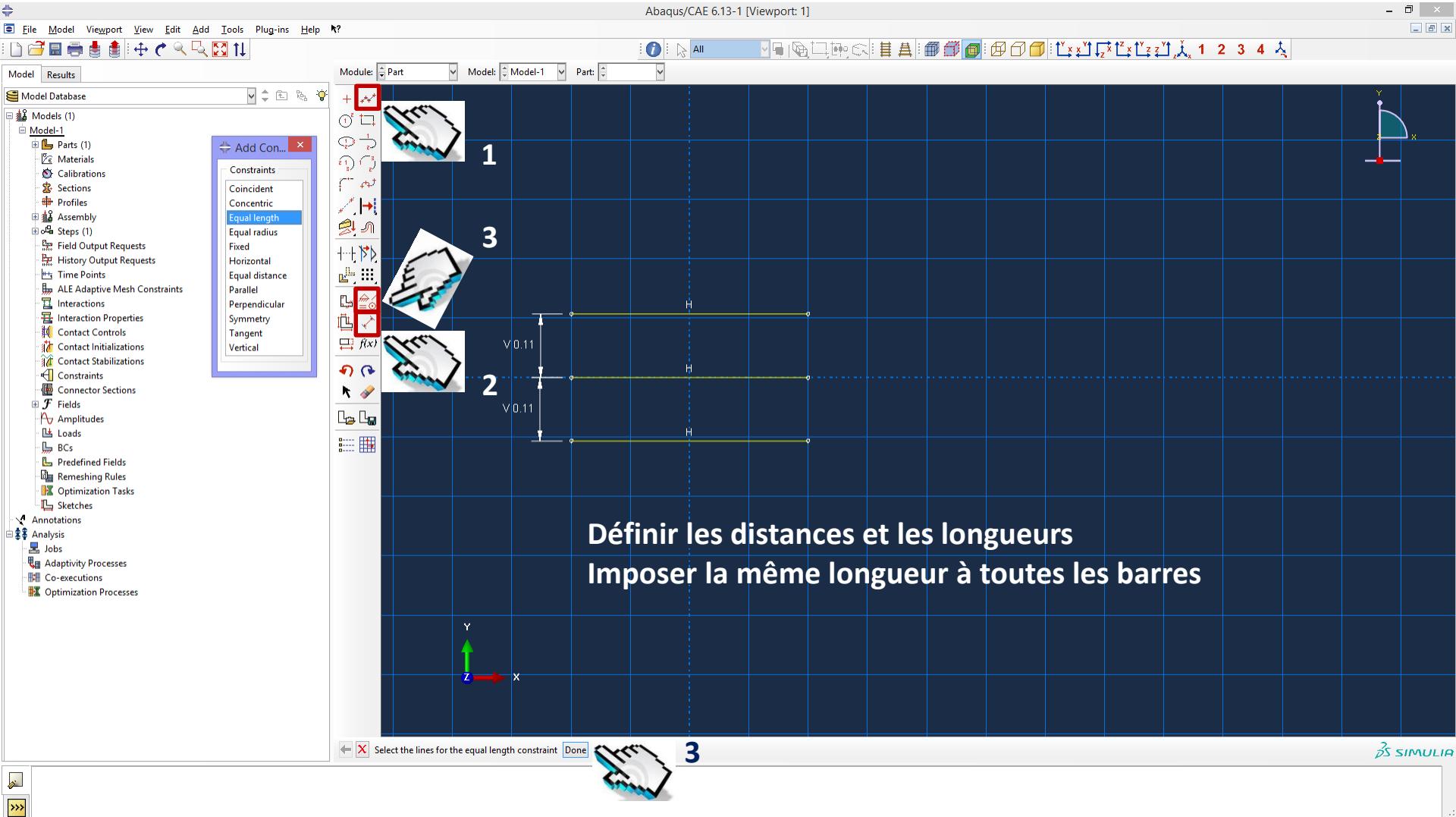
Model - Parts

1. Géométrie : poutre en béton et barres en acier



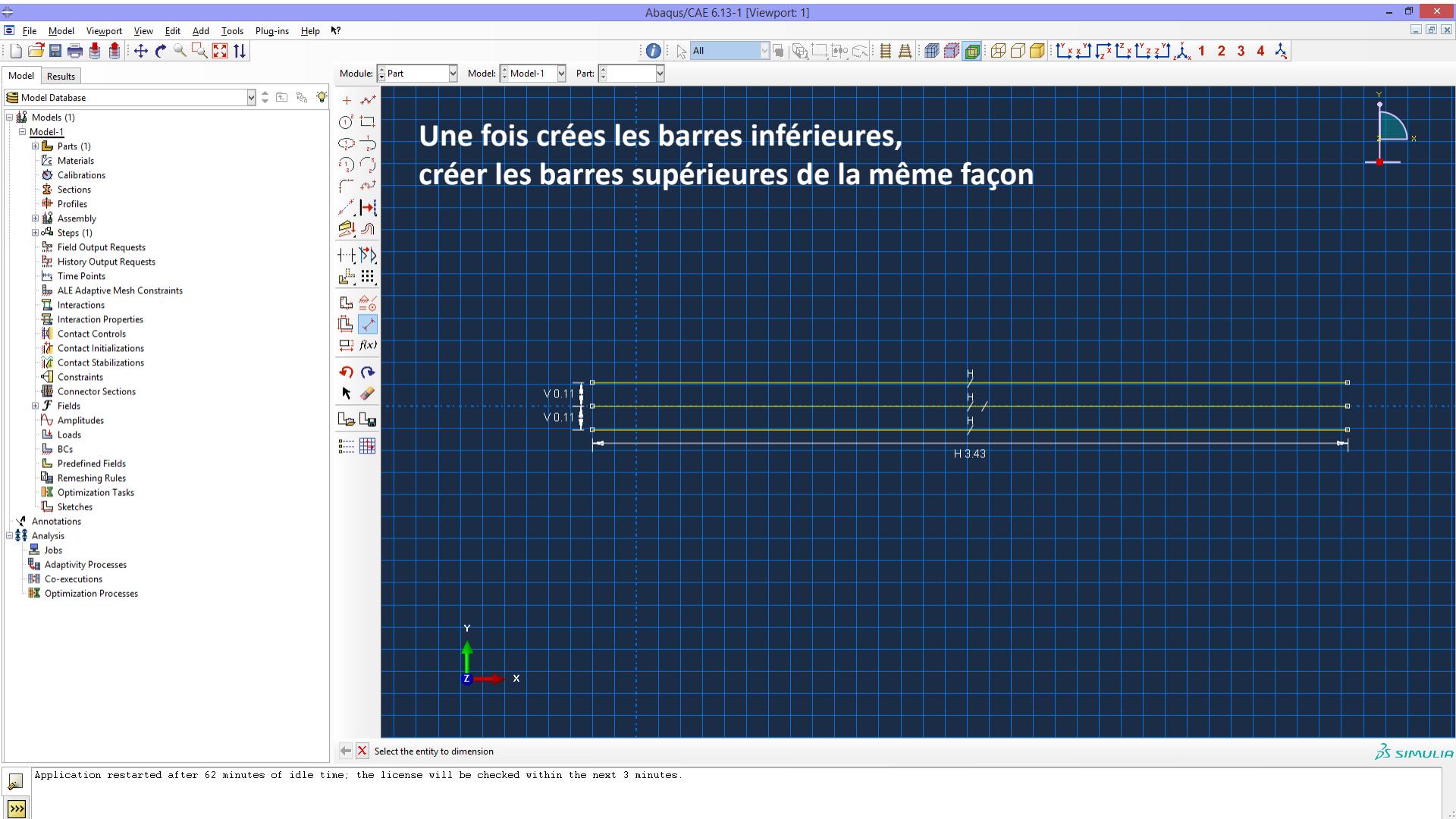
Model - Parts

1. Géométrie : barres longitudinales



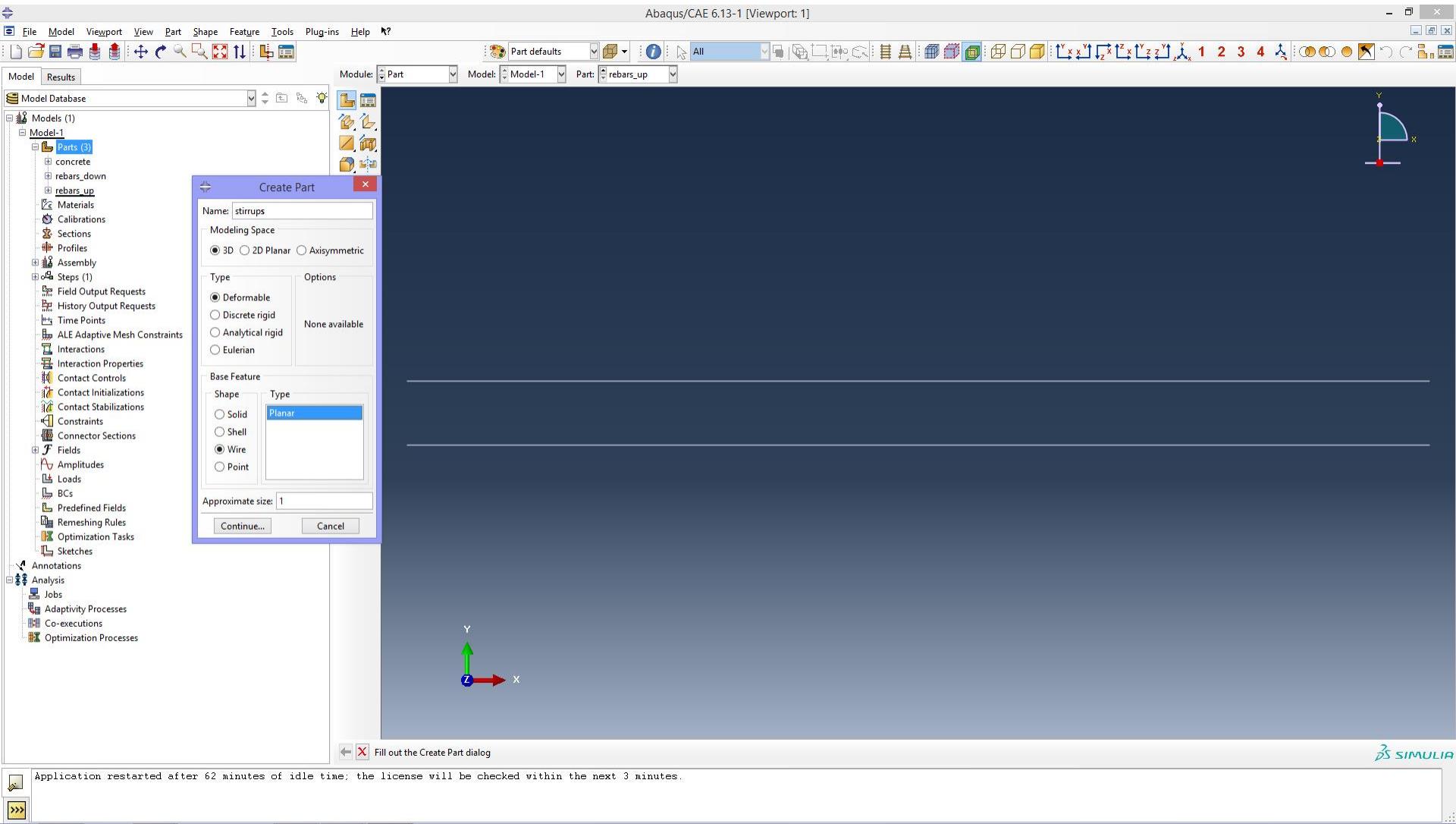
Model - Parts

1. Géométrie : barres longitudinales



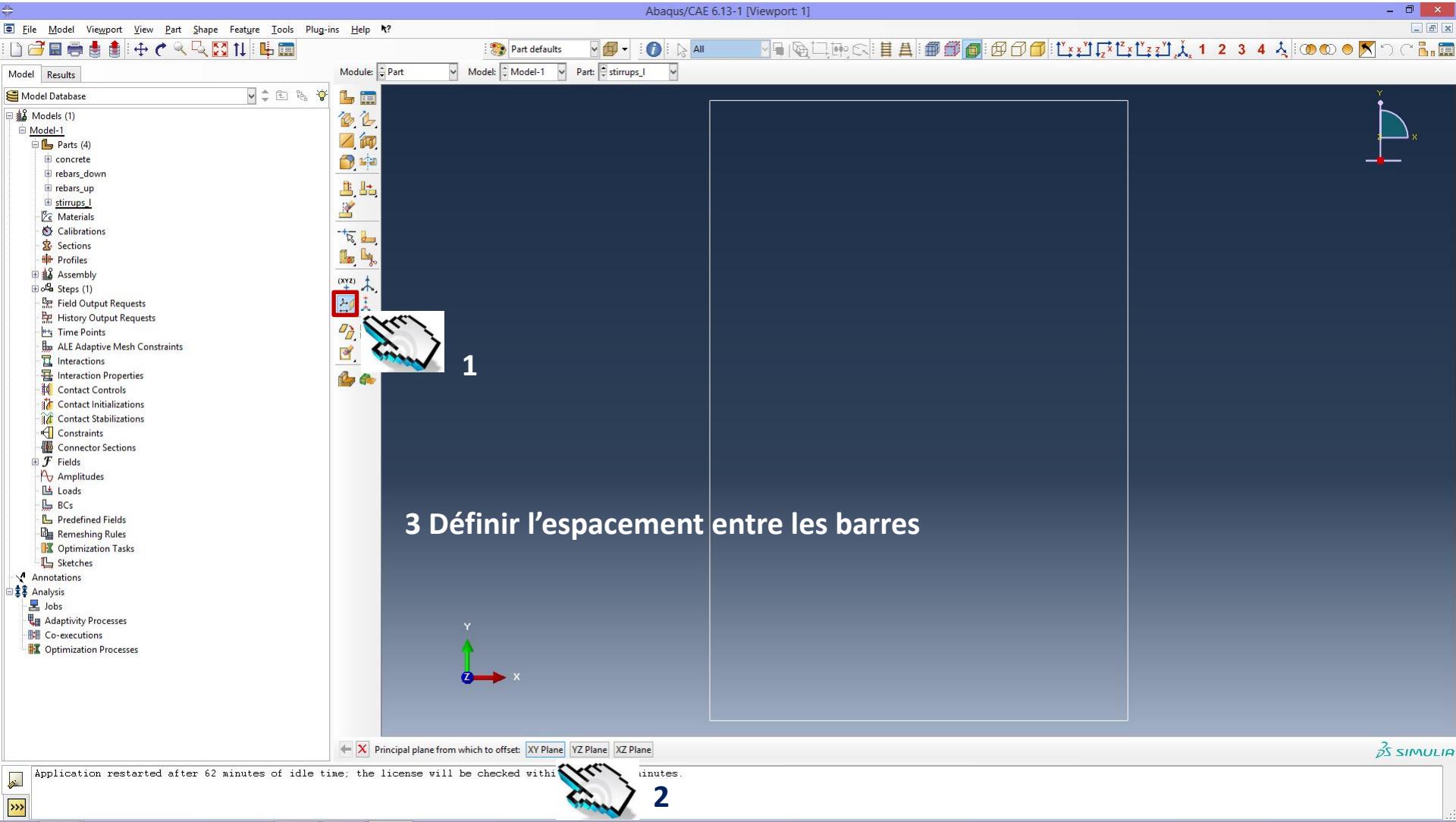
Model - Parts

1. Géométrie : barres transversales



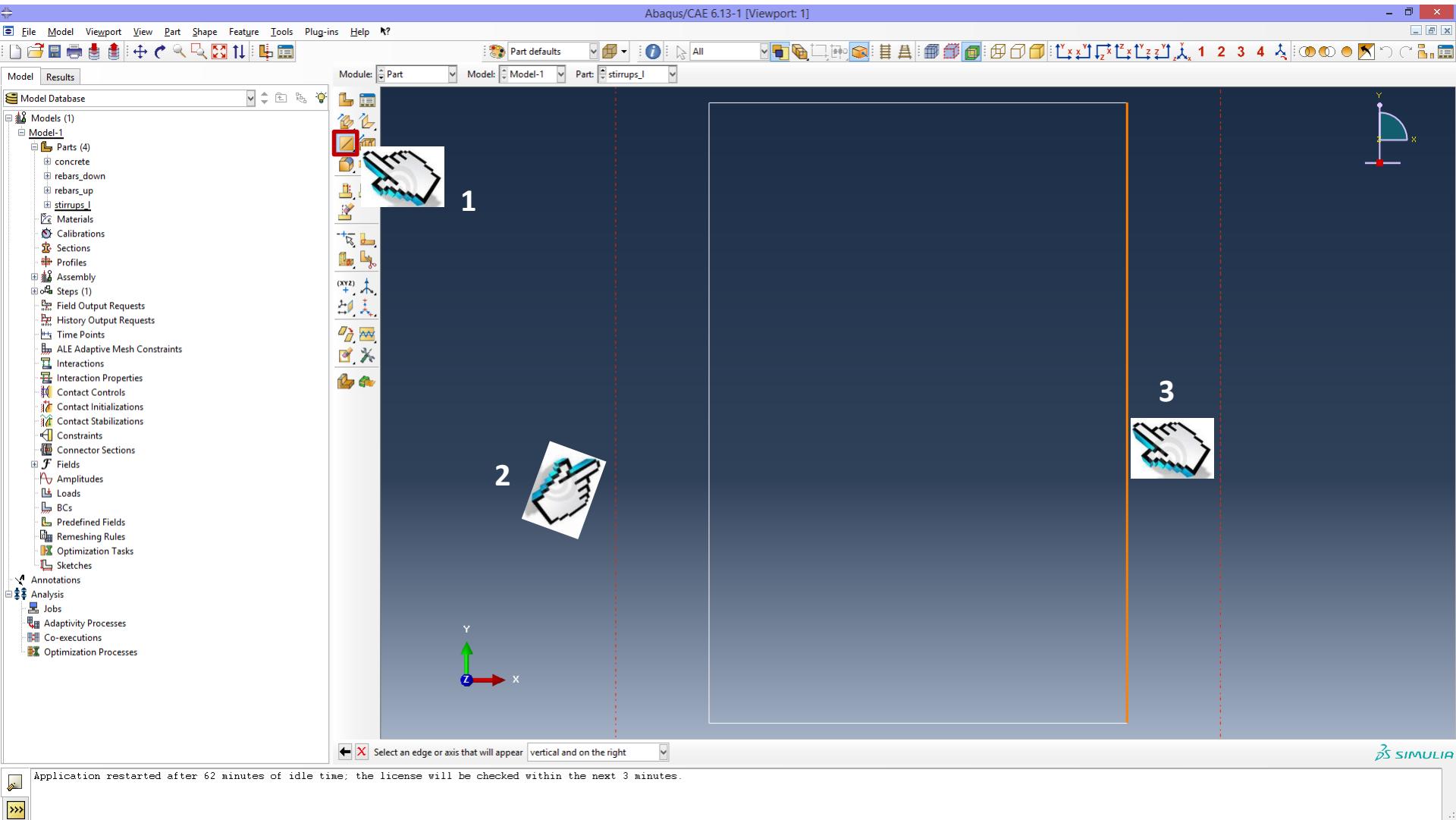
Model - Parts

1. Géométrie : barres transversales



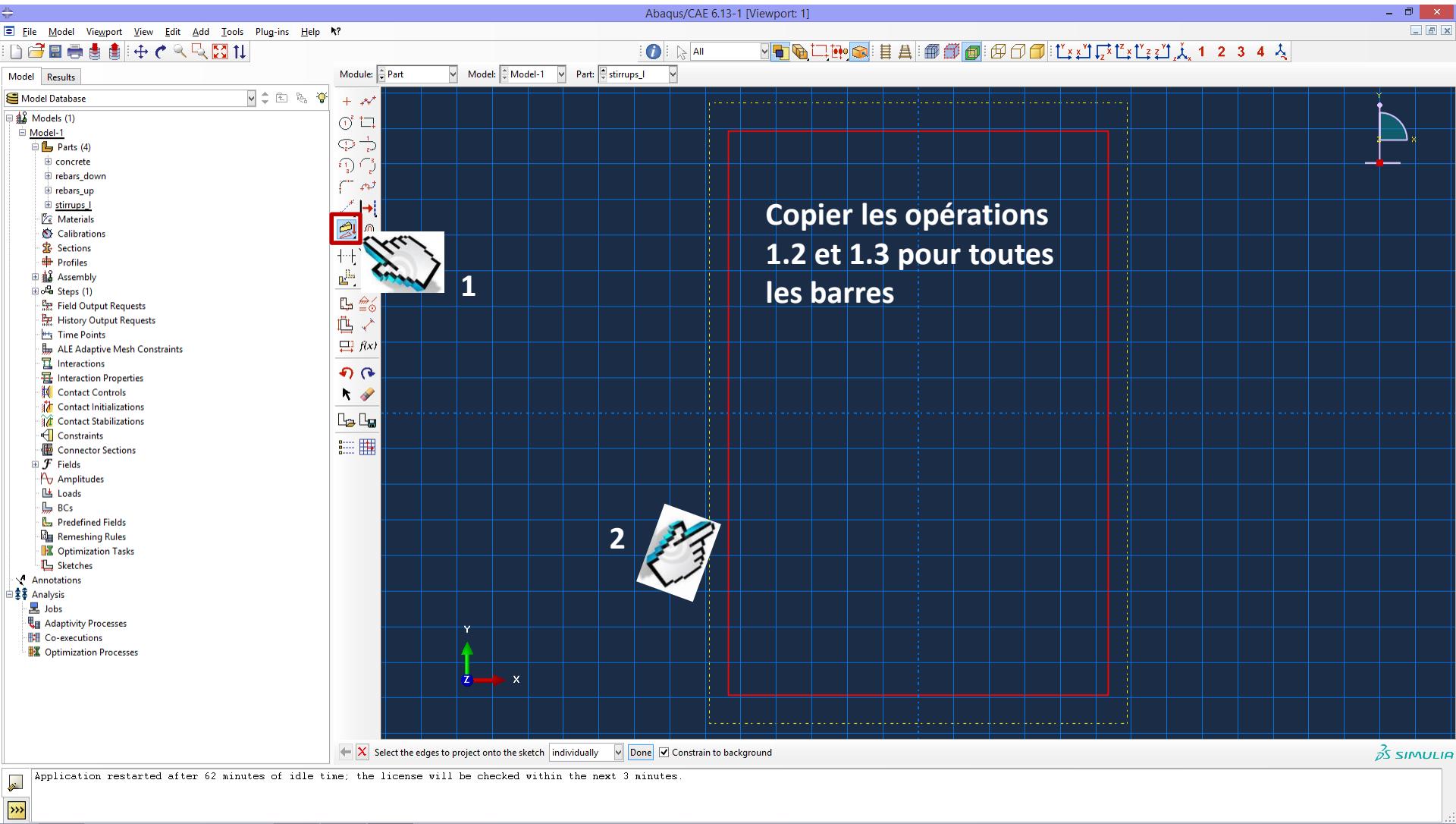
Model - Parts

1.2 Géométrie : créer un nouveau plan où copier la barre transversale



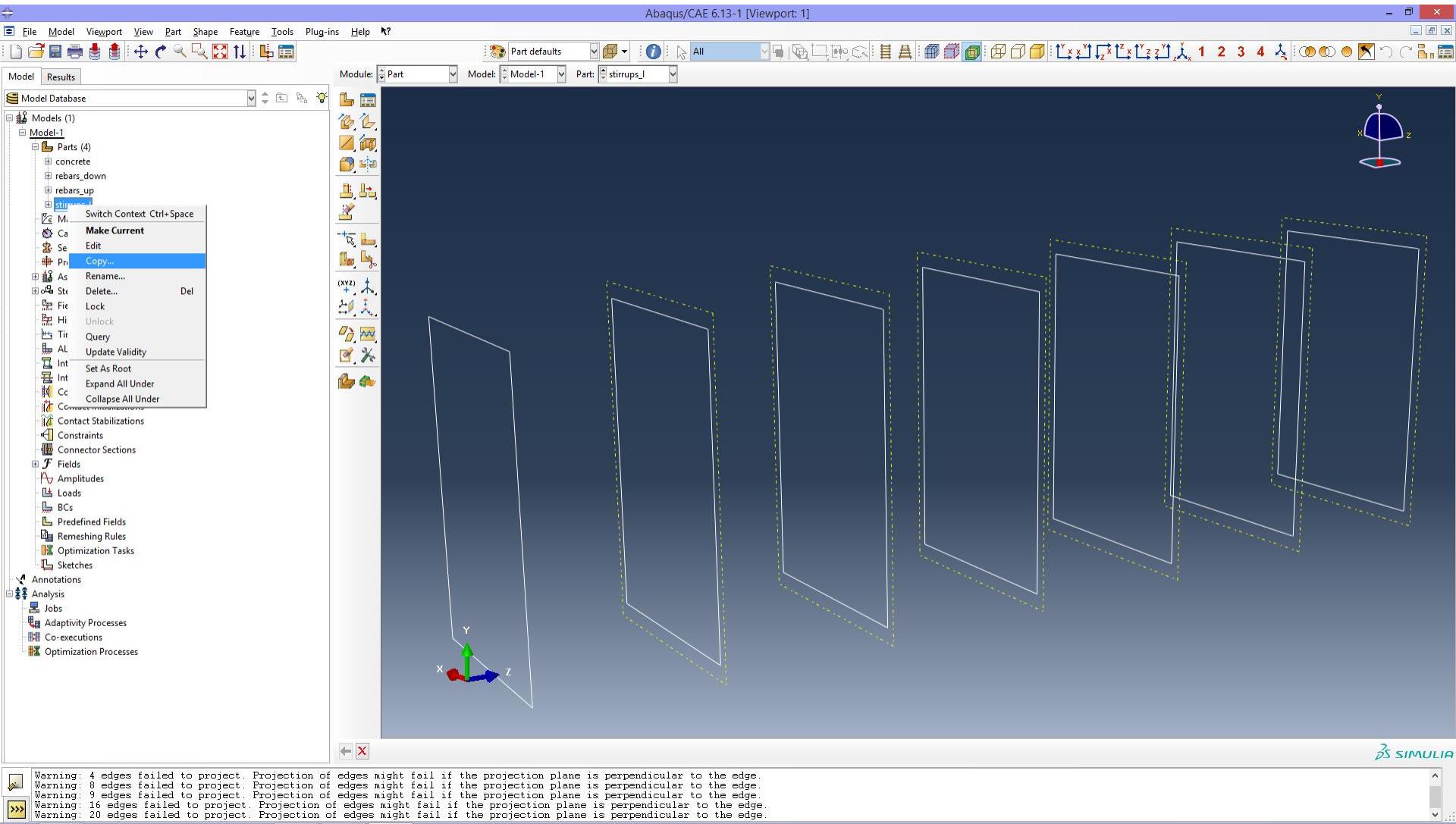
Model - Parts

1.3 Géométrie : copier la barre transversale dans le nouveau plan



Model - Parts

1.4 Géométrie : copier le groupe des barres transversales



Model - Materials

2. Matériaux : densité, module de Young, coefficient de Poisson pour le béton

The screenshot shows the Abaqus/CAE 6.13-1 interface. A modal dialog box titled "Edit Material" is open, showing the properties for a material named "concrete". The "Material Behaviors" section is set to "Elastic". Under the "Elastic" tab, the "Type" is set to "Isotropic". The "Data" table contains one entry: "Young's Modulus" is 31220000000, and "Poisson's Ratio" is 0.2. The background of the interface shows a 3D model of a concrete element with stirrups, and a warning message at the bottom indicates issues with edge projections.

Abaqus/CAE 6.13-1 [Viewport: 1]

Module: Property Model: Model-1 Part: stirrups_r

Model Results Material Library

Model Database

Model-1

Parts (5)

Materials

Calibrations

Sections

Profiles

Assembly

Steps (1)

Field Output Requests

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

Optimization Processes

Edit Material

Name: concrete

Description:

Material Behaviors

Density

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic

Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

No compression

No tension

Data

| | Young's Modulus | Poisson's Ratio |
|---|-----------------|-----------------|
| 1 | 31220000000 | 0.2 |

OK Cancel

Y
Z
X

Unités de mesure du SI : kg, m, s, N

Warning: 4 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 8 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 9 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 16 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 20 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.

SIMULIA

Model - Materials

2. Matériau : densité, élasticité, plasticité pour l'acier

The screenshot shows the Abaqus/CAE 6.13-1 interface. The main window displays a 3D model of four rectangular stirrups, each with a U-shaped notch at one corner. A coordinate system (X, Y, Z) is shown in the bottom left. The top menu bar includes File, Model, Viewport, View, Material, Section, Profile, Composite, Assign, Special, Feature, Tools, Plugins, Help, and a question mark icon. The toolbar below the menu contains various icons for selection, modification, and analysis. The left sidebar is the 'Model Database' tree view, showing the project structure: Model-1, Parts (5), Materials (1), Calibrations, Sections, Profiles, Assembly, Steps (1), Field Output Requests, 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, and Sketches. The 'Materials' node is expanded, showing one material entry: 'acier'. The 'Edit Material' dialog box is open in the center, titled 'Edit Material'. It has a 'Name' field set to 'acier' and a 'Description' field that is empty. Under 'Material Behaviors', 'Plastic' is selected. Below it are tabs for General, Mechanical, Thermal, Electrical/Magnetic, and Other. The 'Plastic' tab is active, showing 'Hardening: Isotropic' selected. There are two checkboxes: 'Use strain-rate-dependent data' and 'Use temperature-dependent data', both of which are unchecked. The 'Number of field variables:' field is set to 0. The 'Data' section contains a table:

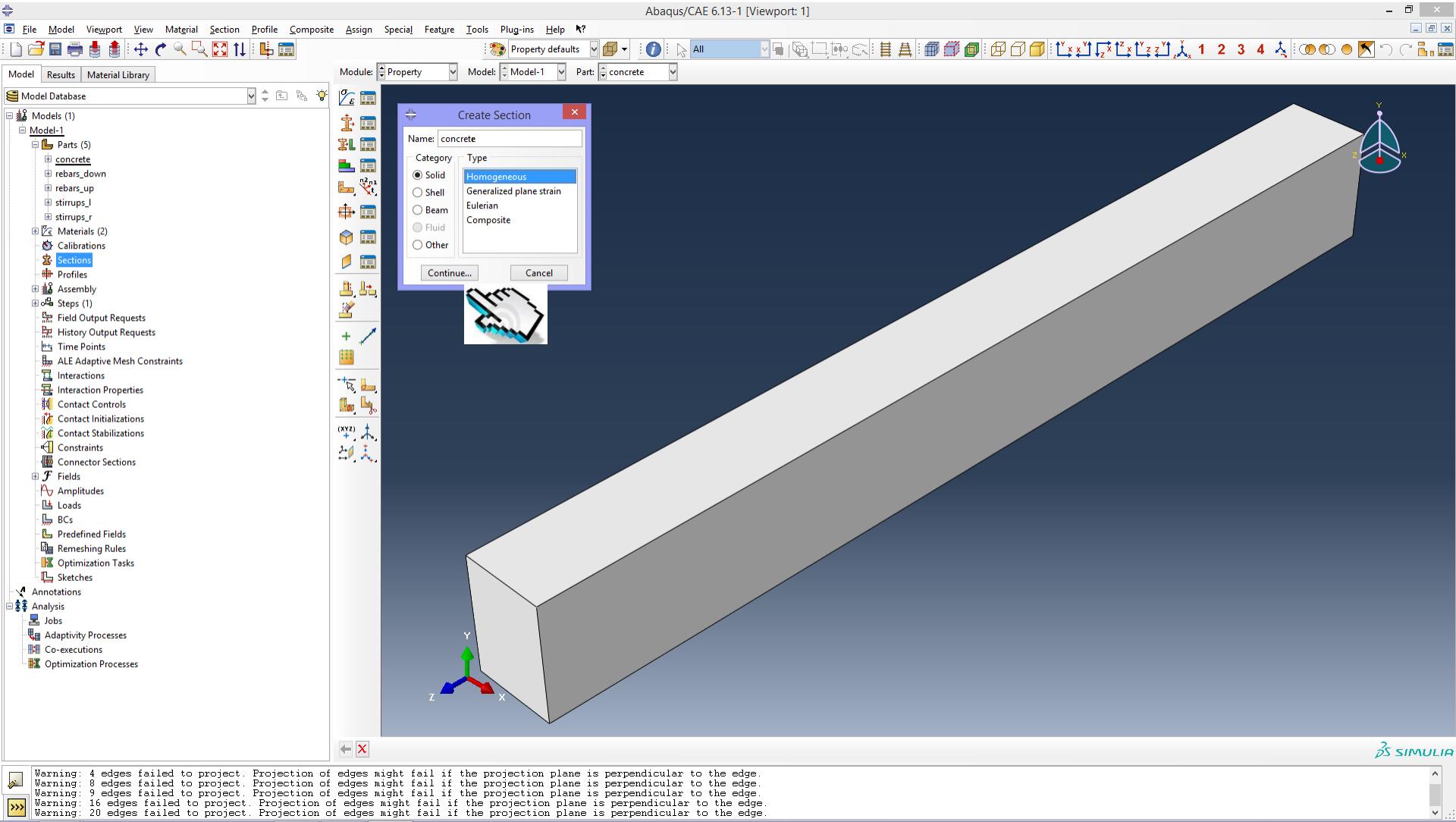
| | Yield Stress | Plastic Strain |
|---|--------------|----------------|
| 1 | 450000000 | 0 |
| 2 | 540000000 | .098 |

At the bottom of the dialog are 'OK' and 'Cancel' buttons. A small coordinate system (X, Y, Z) is also present at the bottom left of the dialog. The bottom status bar displays several warning messages about edge projections failing to project.

Warning: 4 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 8 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 9 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 16 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 20 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.

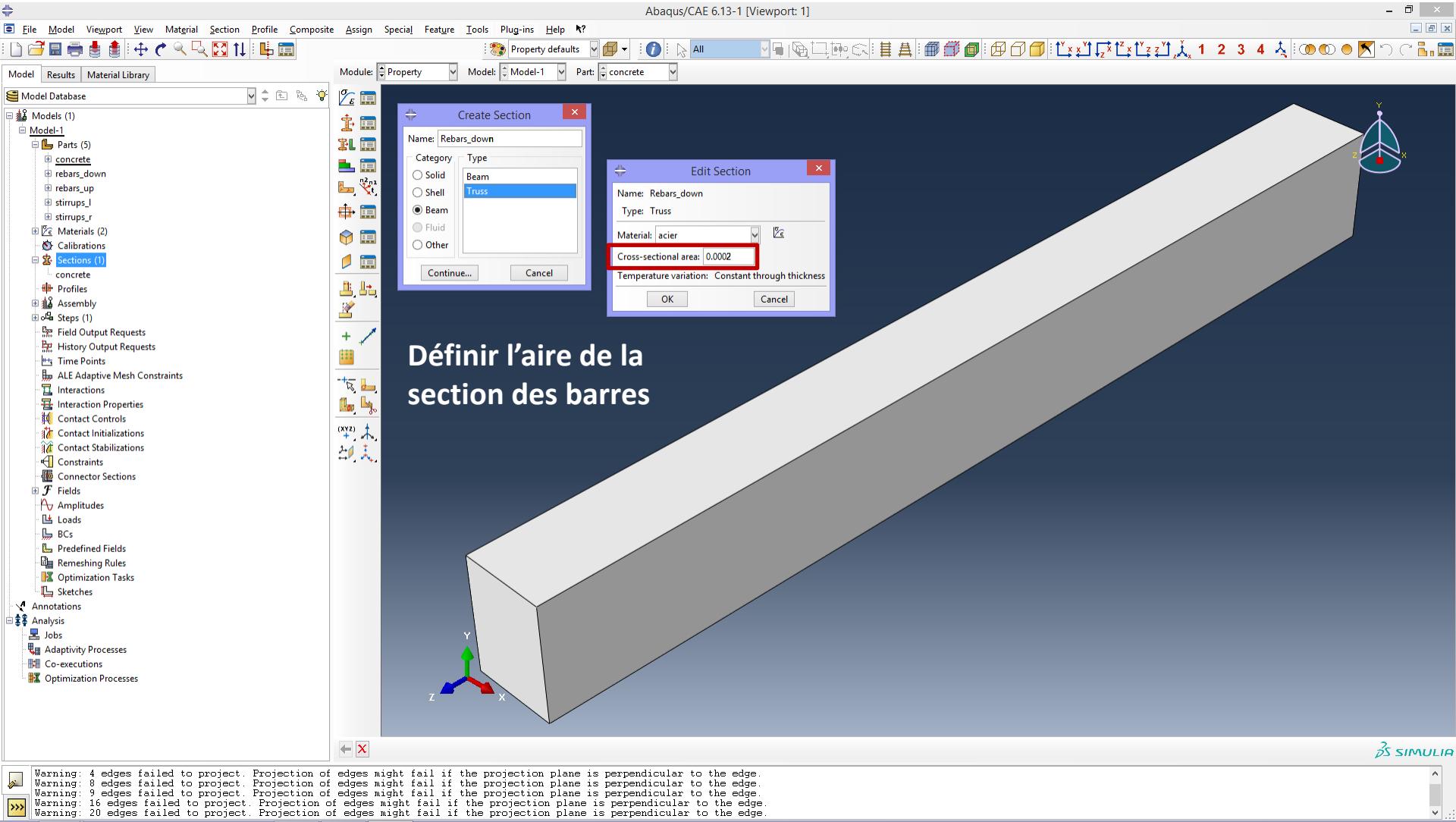
Model - Sections

3. Section : type de modèle et type de matériau



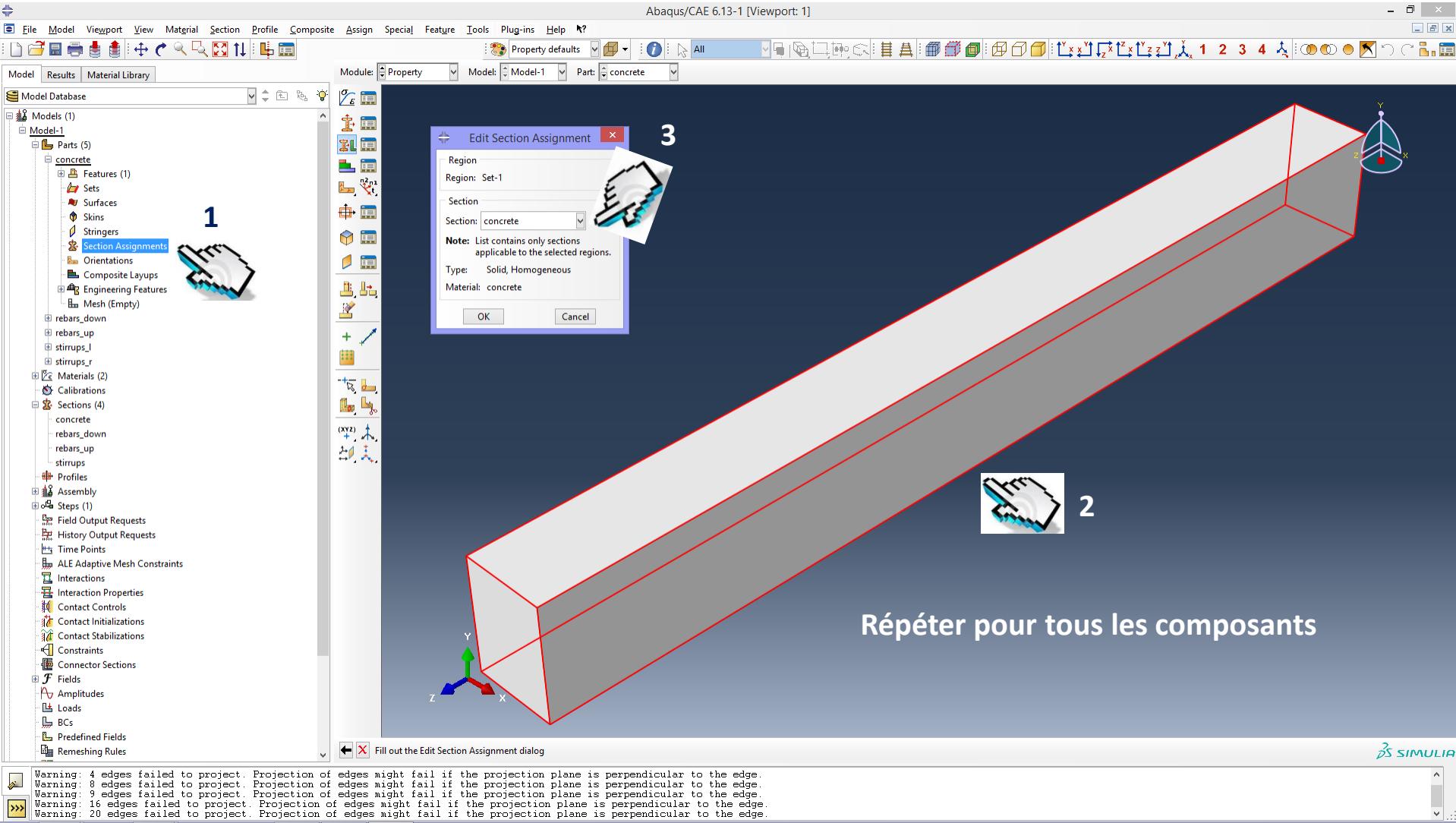
Model - Sections

3. Section : type de modèle et type de matériau



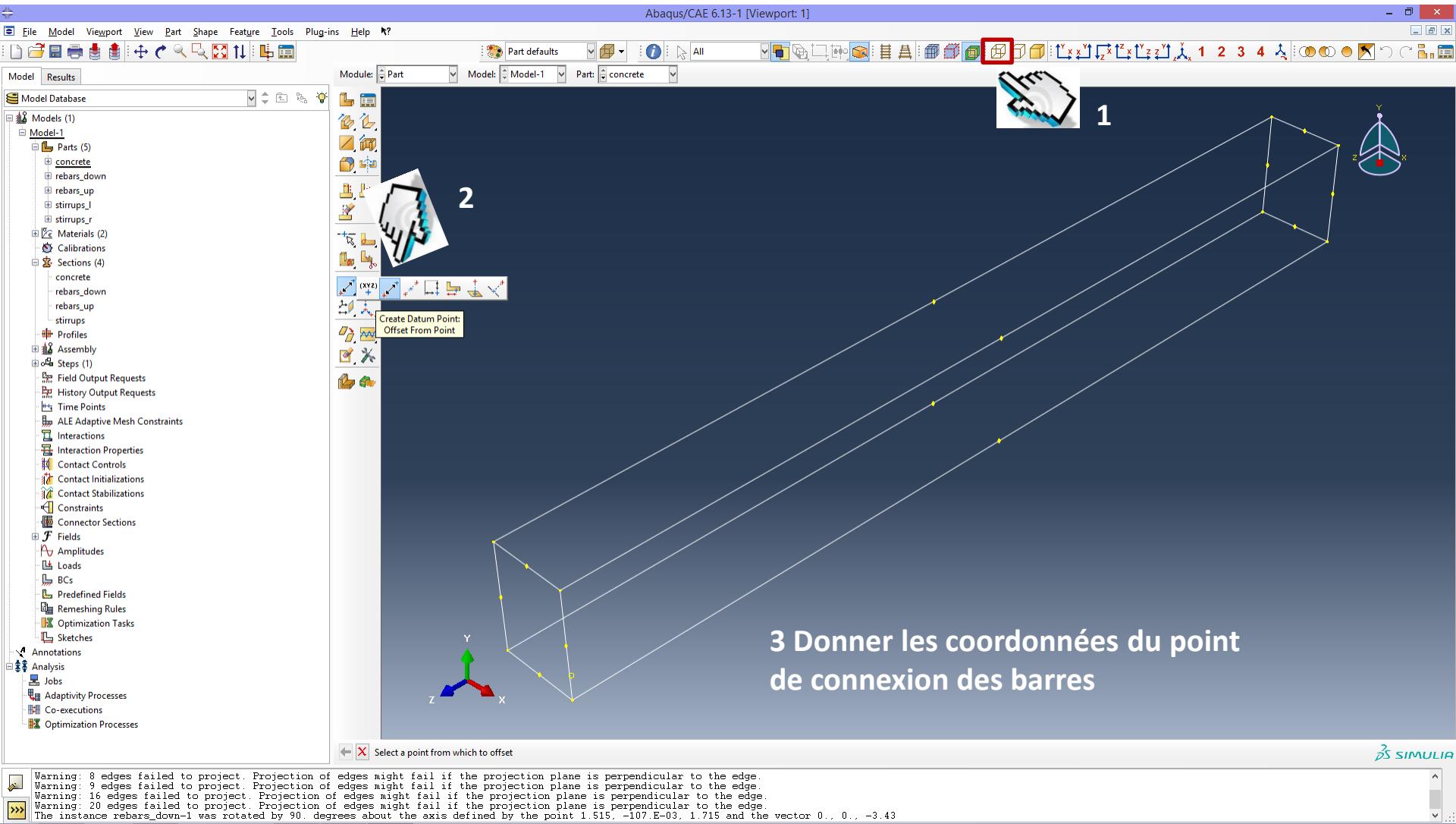
Model - Parts

4. Attribution de la section



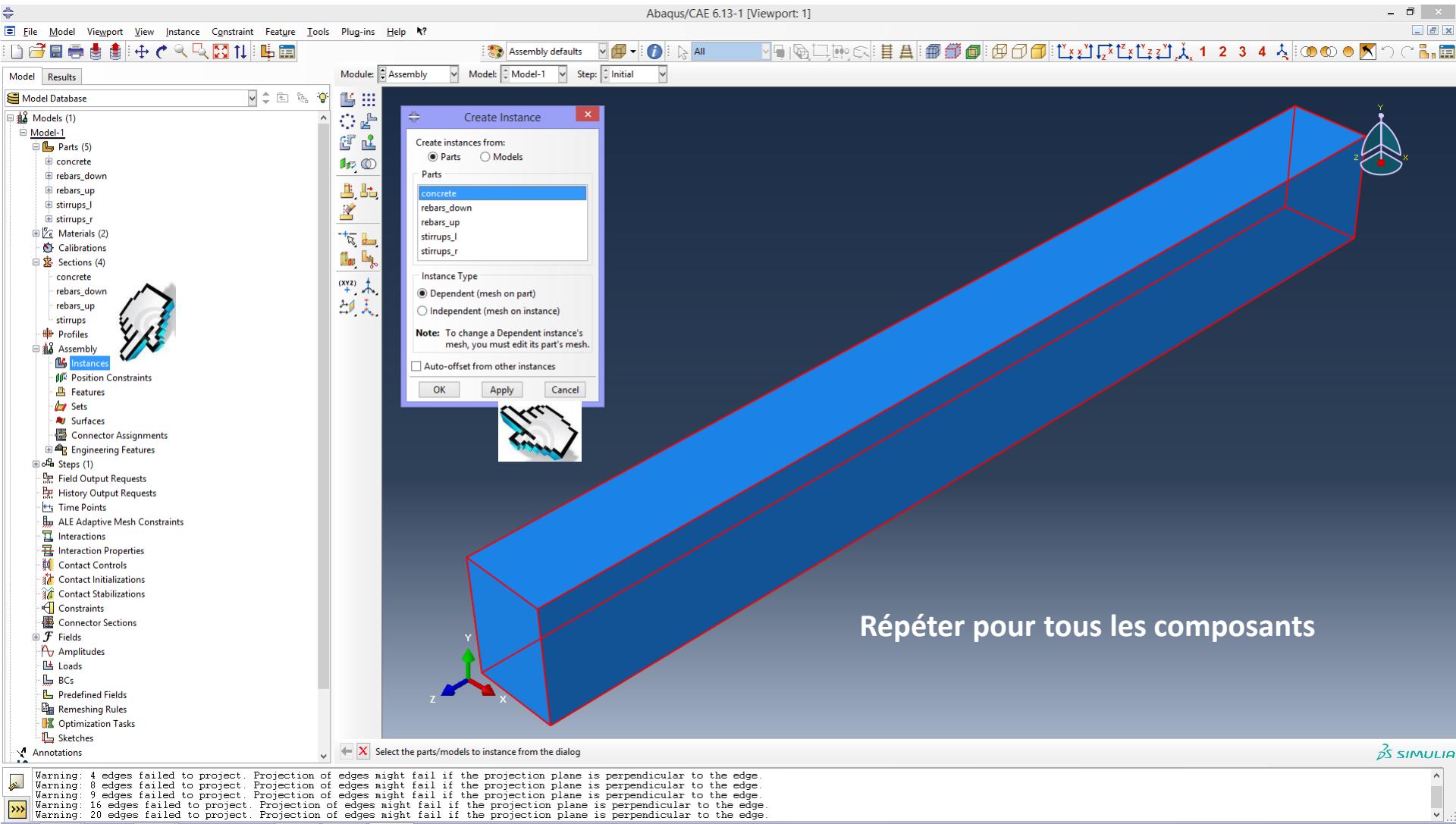
Model - Parts

5. Points de connexion avec les autres composants



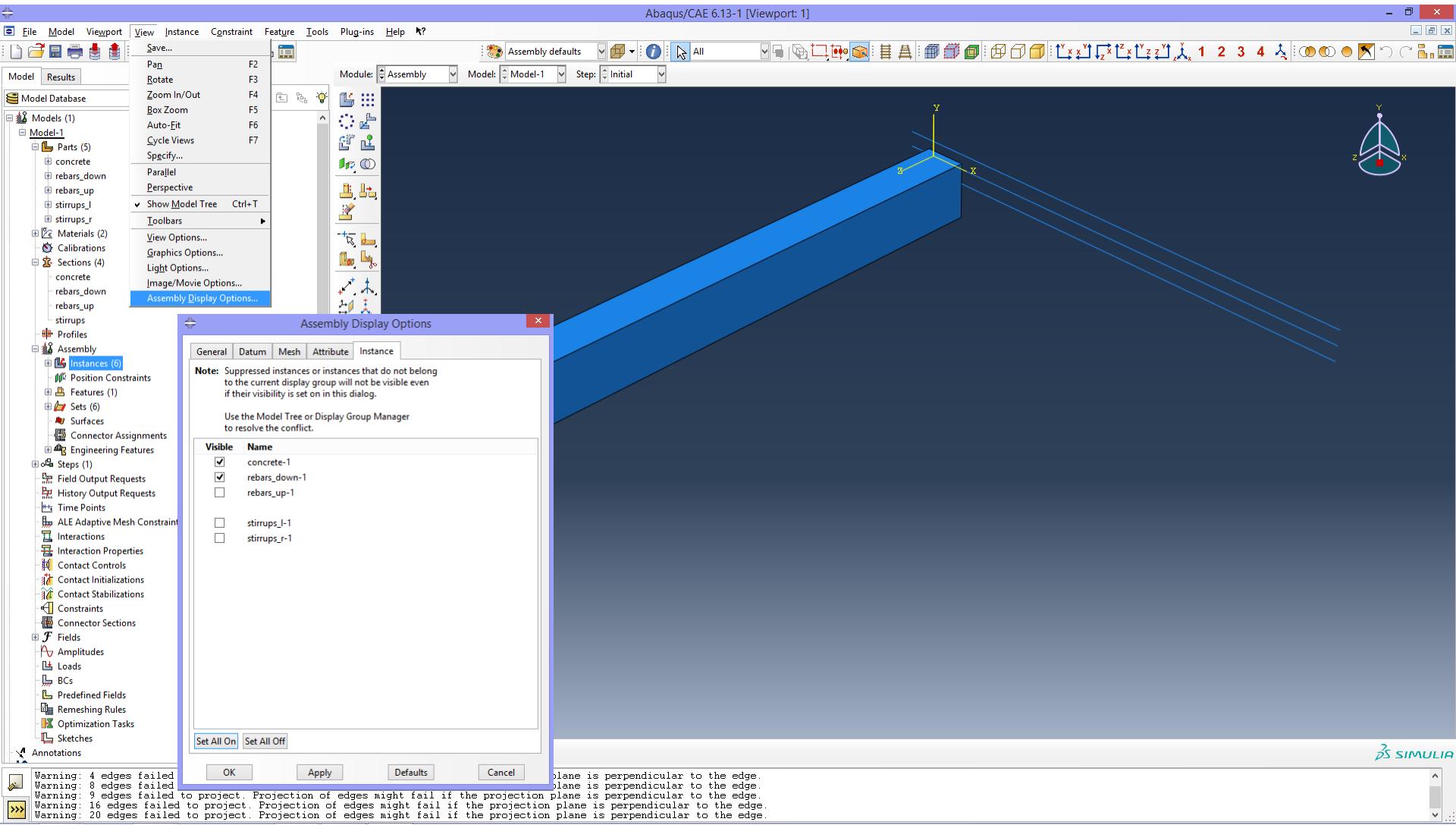
Model - Assembly

6. Assemblage



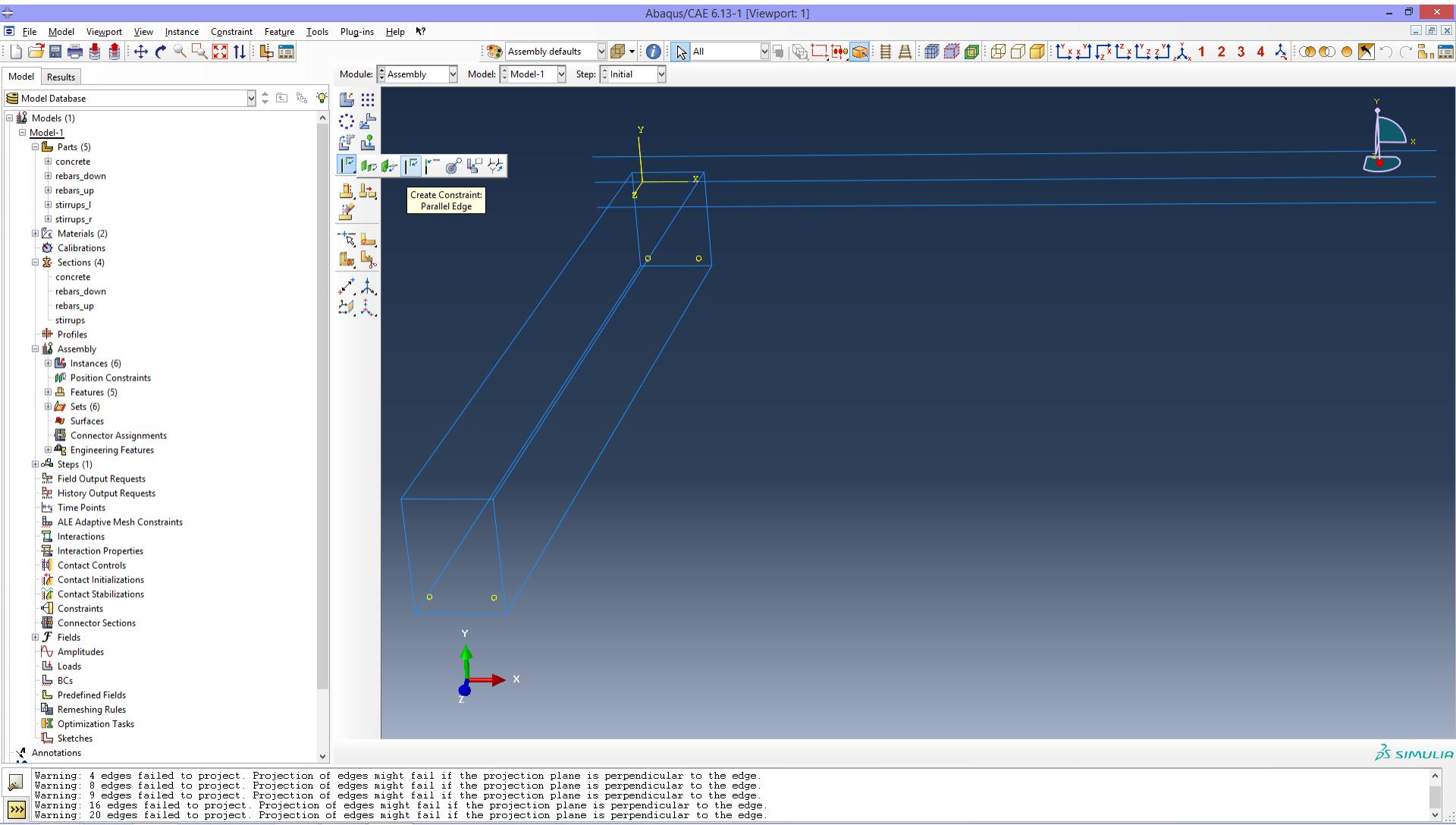
Model - Assembly

6.1 Assemblage



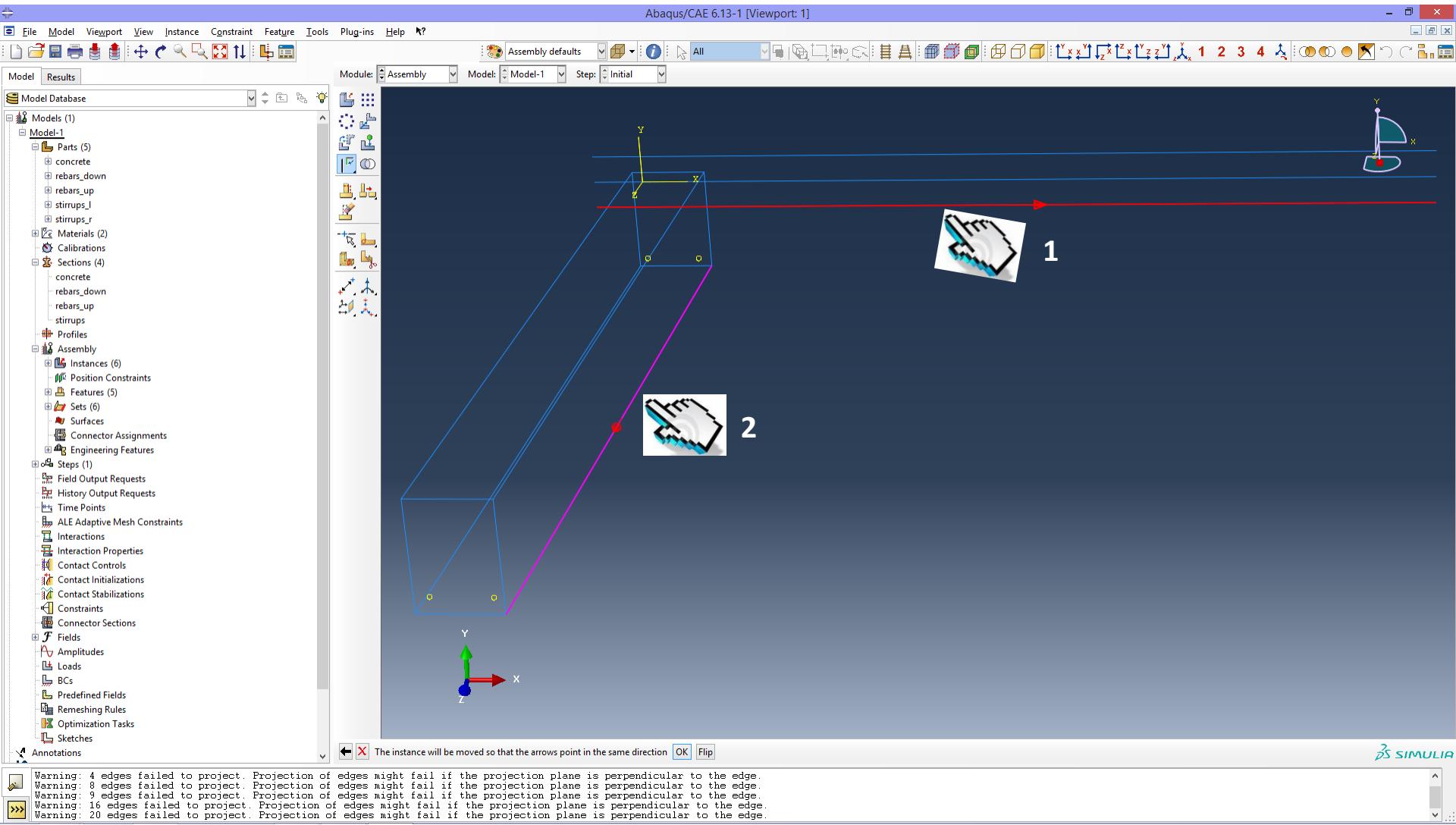
Model - Assembly

6.2 Assemblage



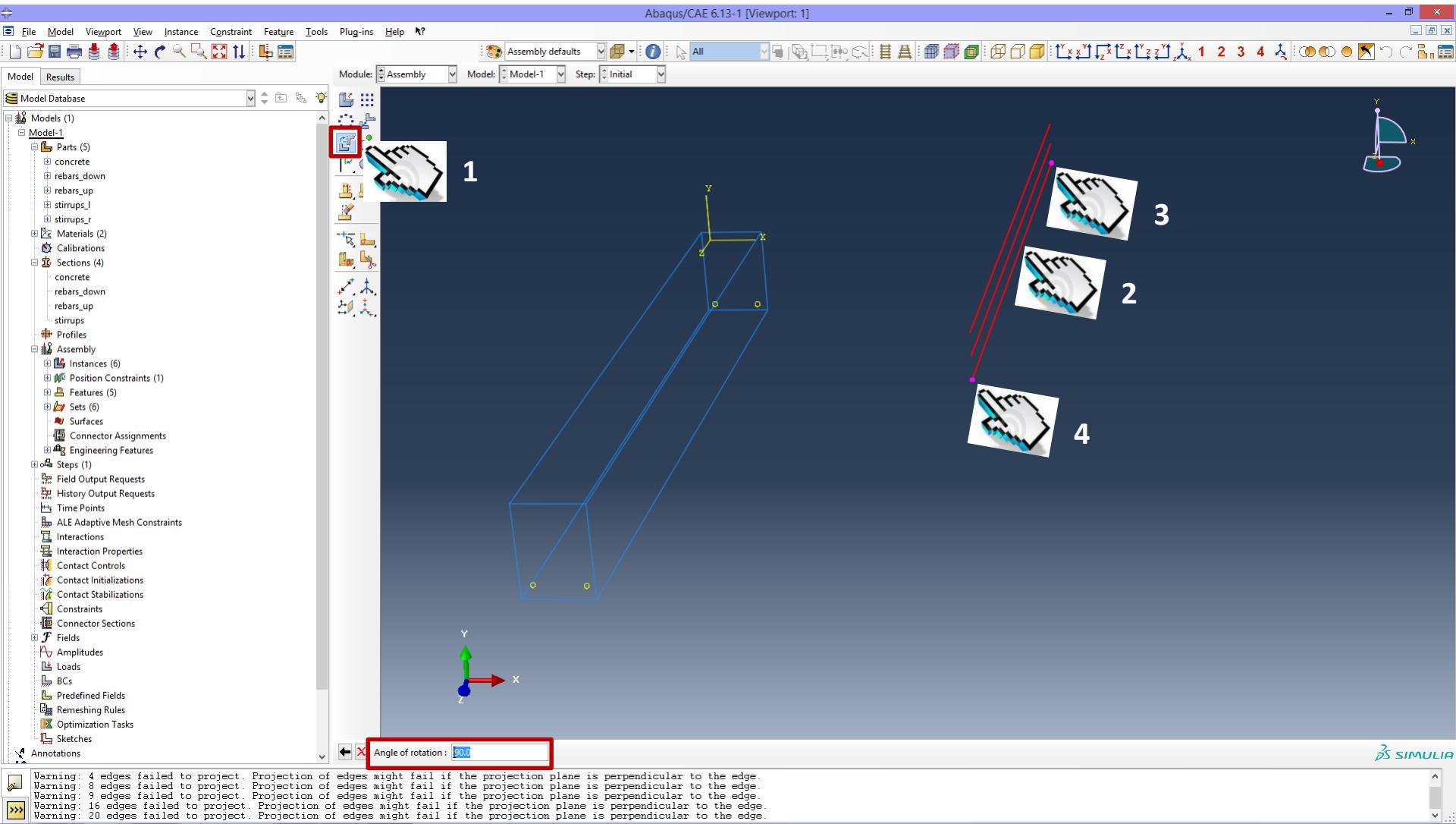
Model - Assembly

6.3 Assemblage



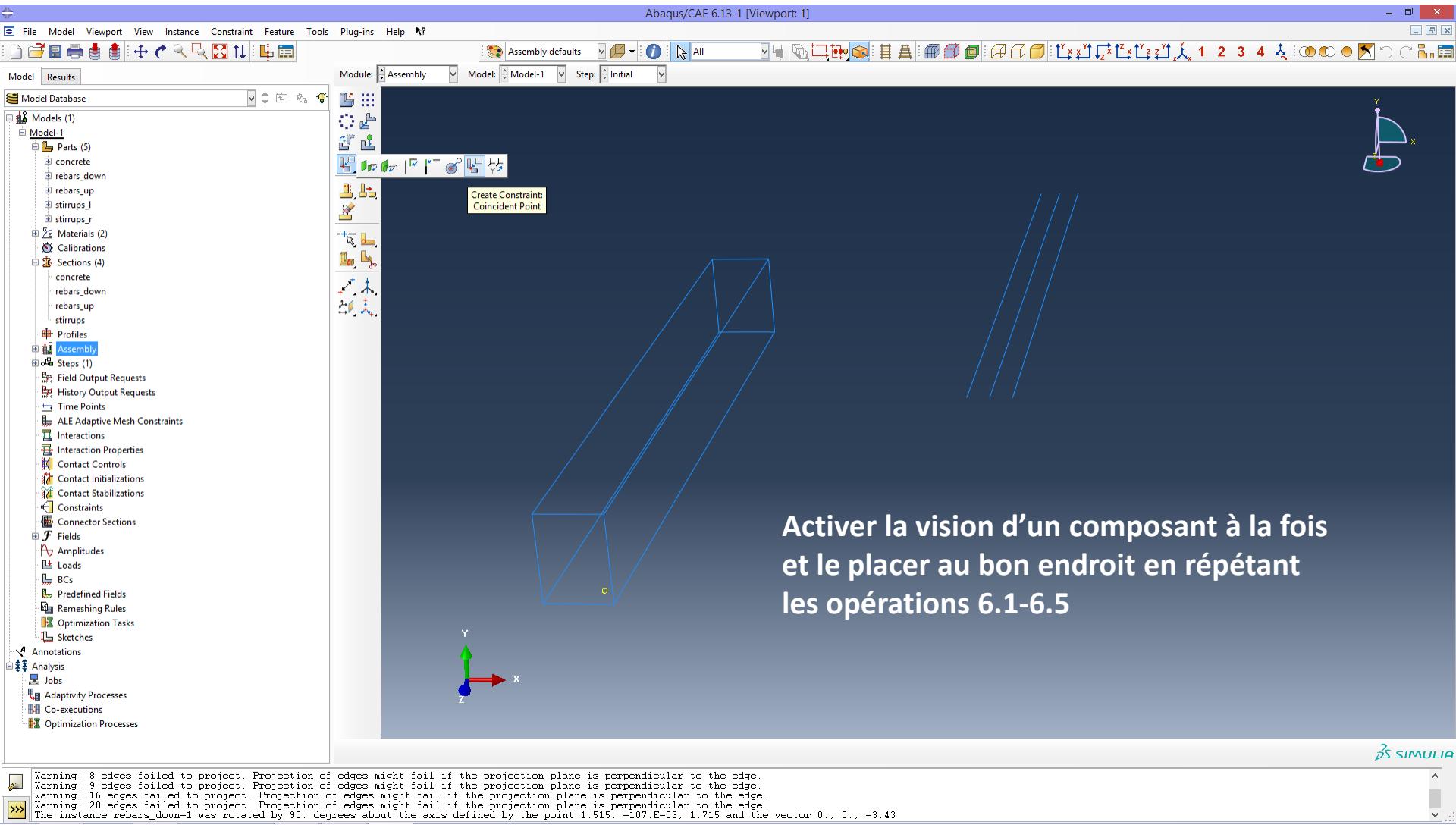
Model - Assembly

6.4 Assemblage



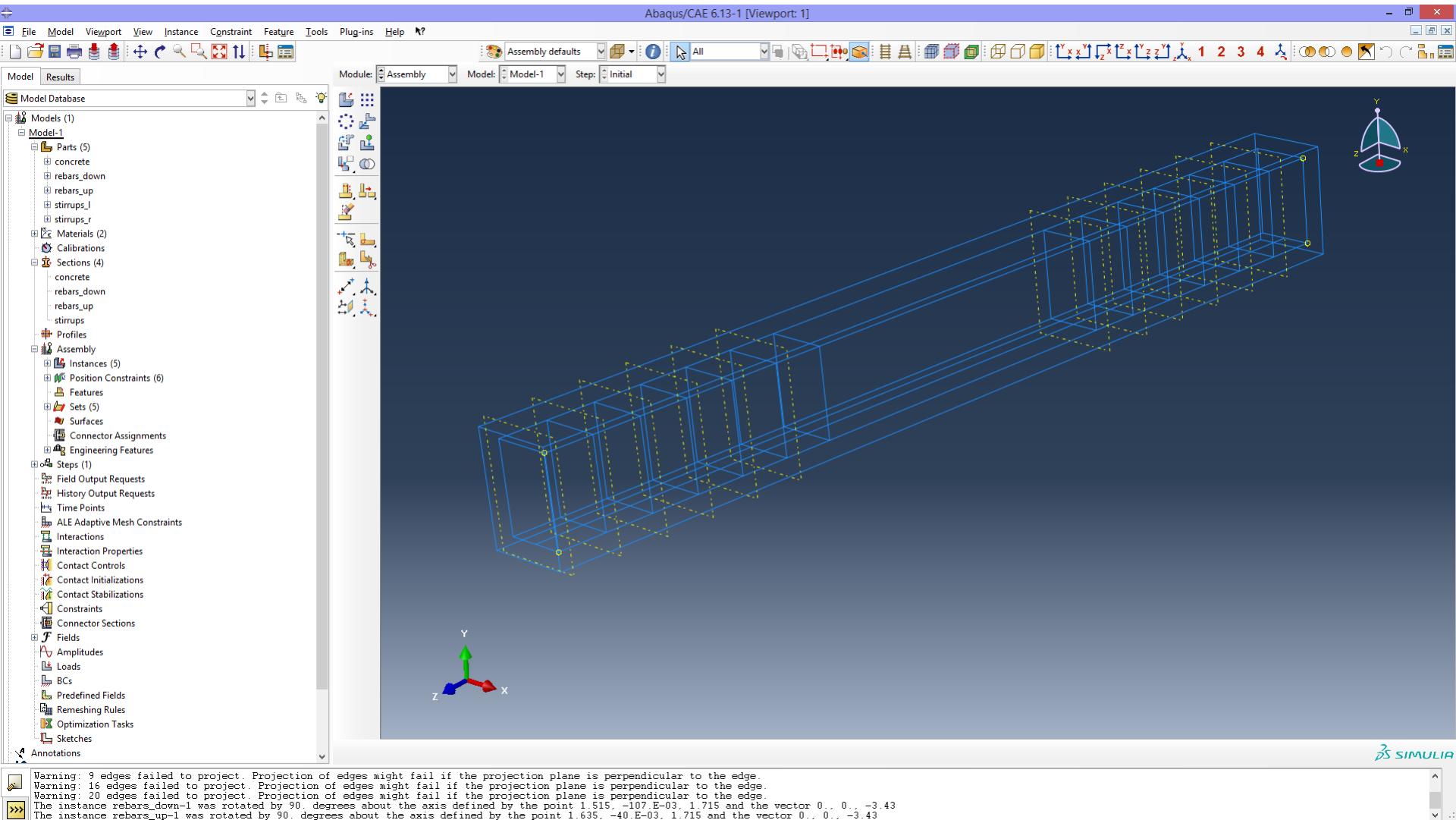
Model - Assembly

6.5 Assemblage



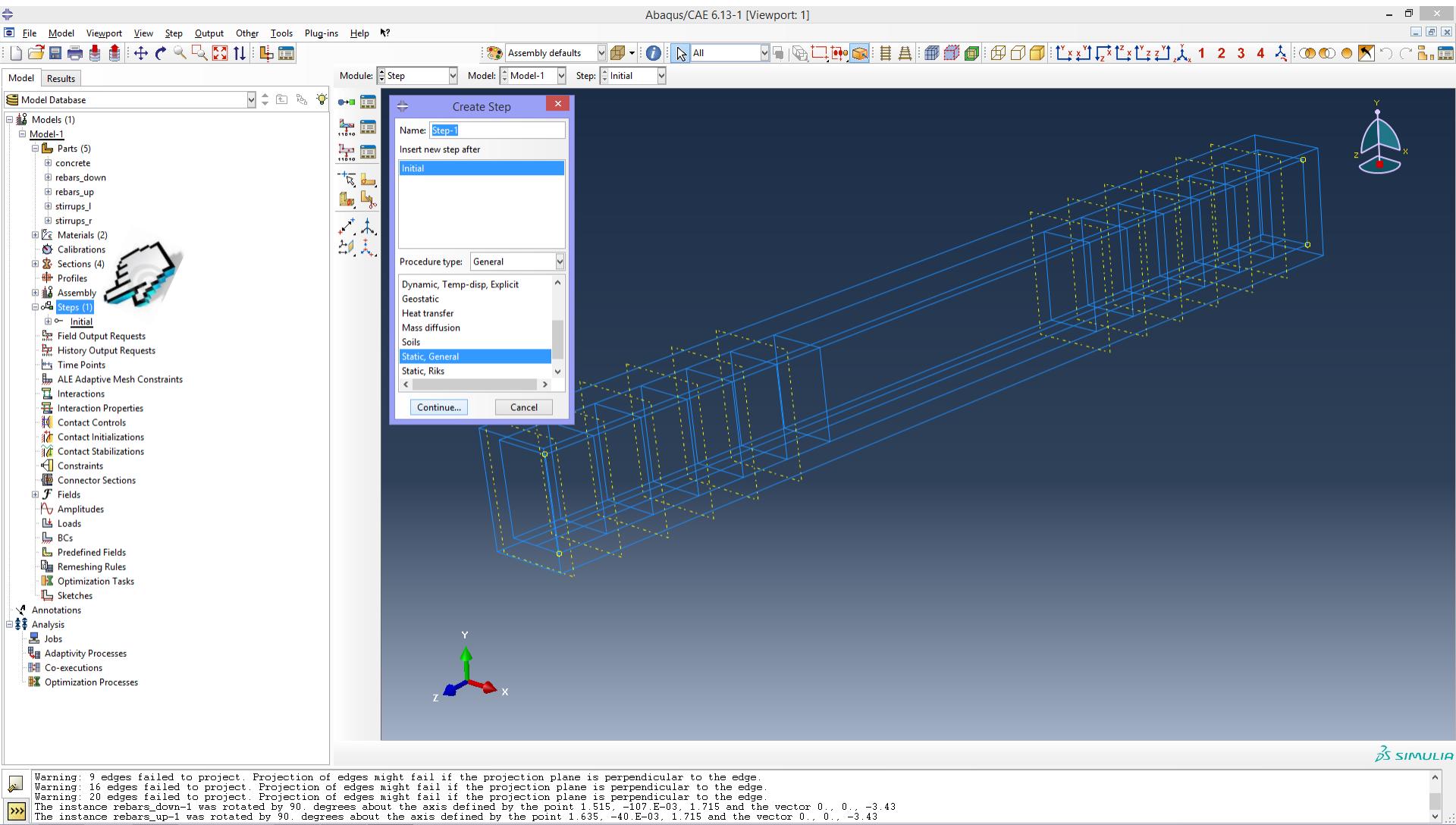
Model - Assembly

6. Assemblage fini



Model - Steps

7. Pas de calcul : Statique, Quasi-statique



Model - Steps

7. Pas de calcul : Statique, Quasi-statique

The screenshot shows the Abaqus/CAE 6.13-1 interface. The left sidebar displays the Model Database with various components like Parts, Materials, Sections, and Steps. The main workspace shows a 3D model of a concrete beam with a stirrup reinforcement. A callout points to the 'Edit Step' dialog box, which is open and set to 'Step-1'. The dialog shows the 'Basic' tab selected, with 'Type: Static, General' and 'Type: Automatic' checked. It also shows 'Maximum number of increments: 100000' and increment size settings. Below the dialog, a small 3D view shows a single element with a coordinate system. The bottom status bar provides warning messages about edge projections and rotated rebars.

Edit Step

Name: Step-1
Type: Static, General

Type: Automatic Fixed

Maximum number of increments: 100000

Initial Minimum Maximum
Increment size: 0.001 1E-09 0.01

OK Cancel

Basic : Time period = durée tmax

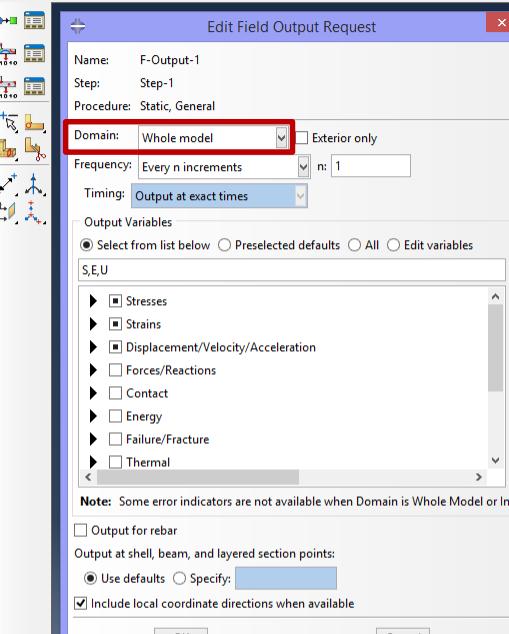
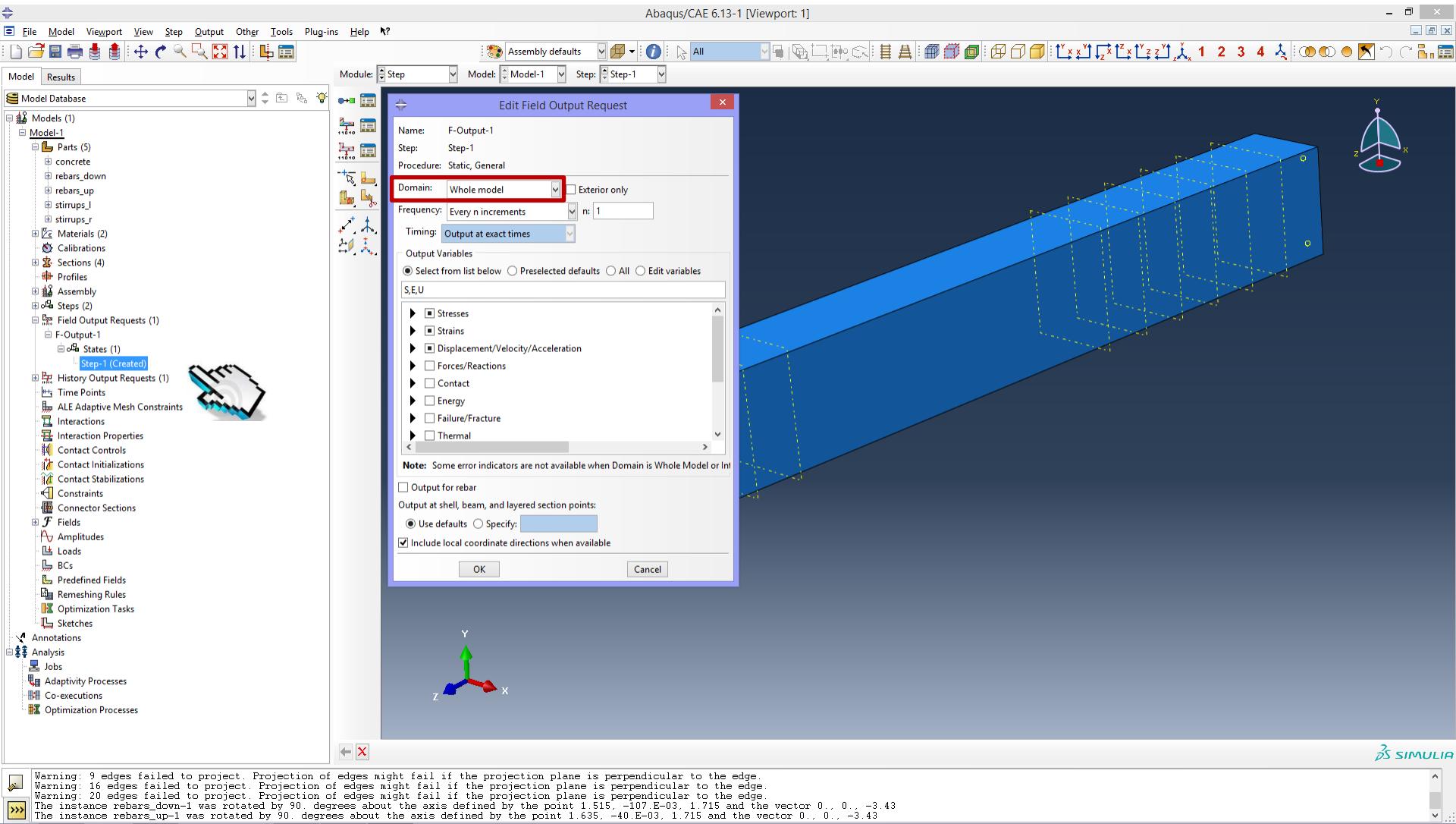
Incrementation : pas de temps dt

Max num of increments >= tmax / dt

Warning: 9 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 16 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
Warning: 20 edges failed to project. Projection of edges might fail if the projection plane is perpendicular to the edge.
The instance rebars_down-1 was rotated by 90. degrees about the axis defined by the point 1.515, -107.E-03, 1.715 and the vector 0., 0., -3.43
The instance rebars_up-1 was rotated by 90. degrees about the axis defined by the point 1.635, -40.E-03, 1.715 and the vector 0., 0., -3.43

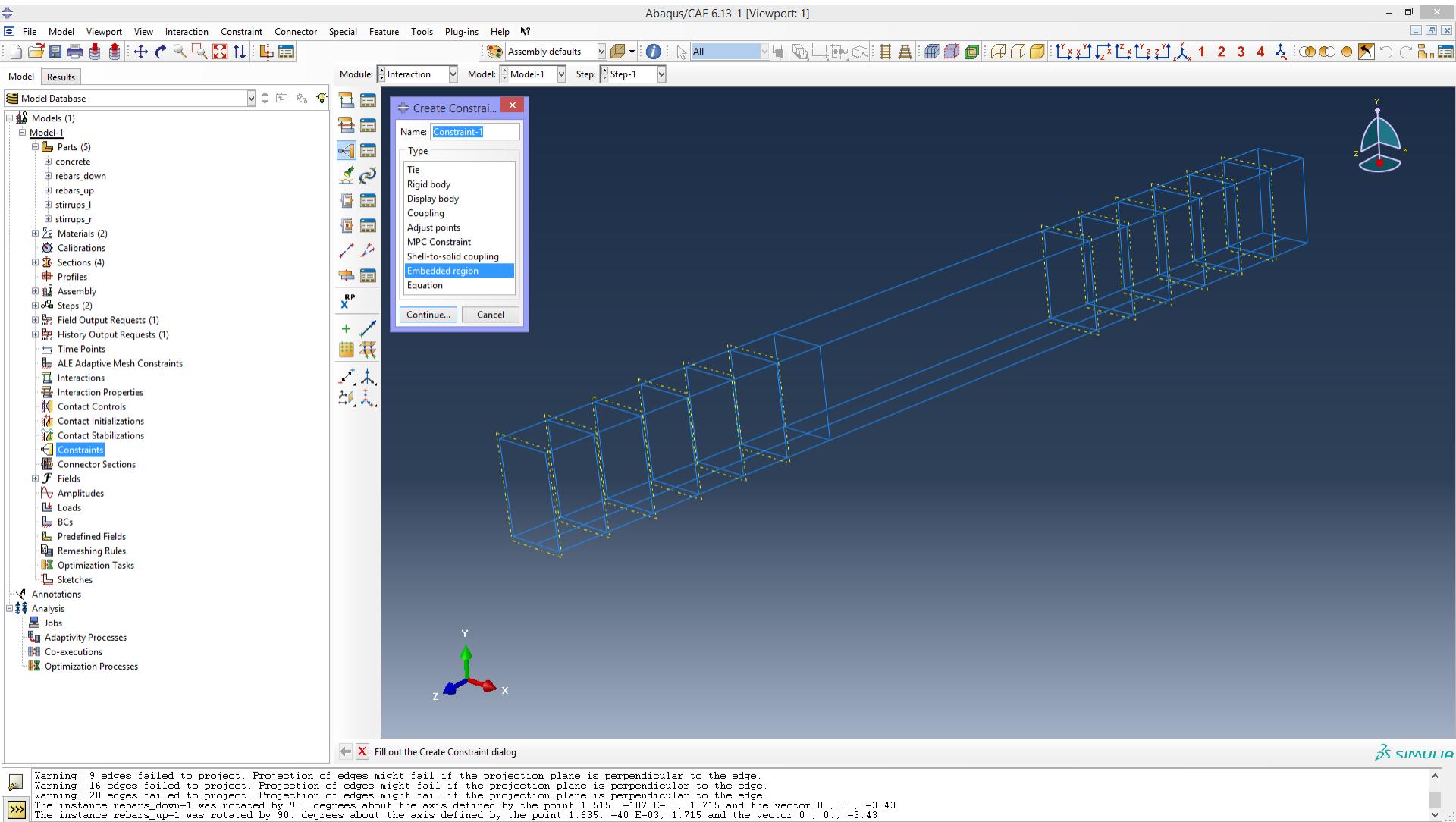
Model - Field Output Requests

8. Résultats demandés : contraintes, déformations totales, déplacements



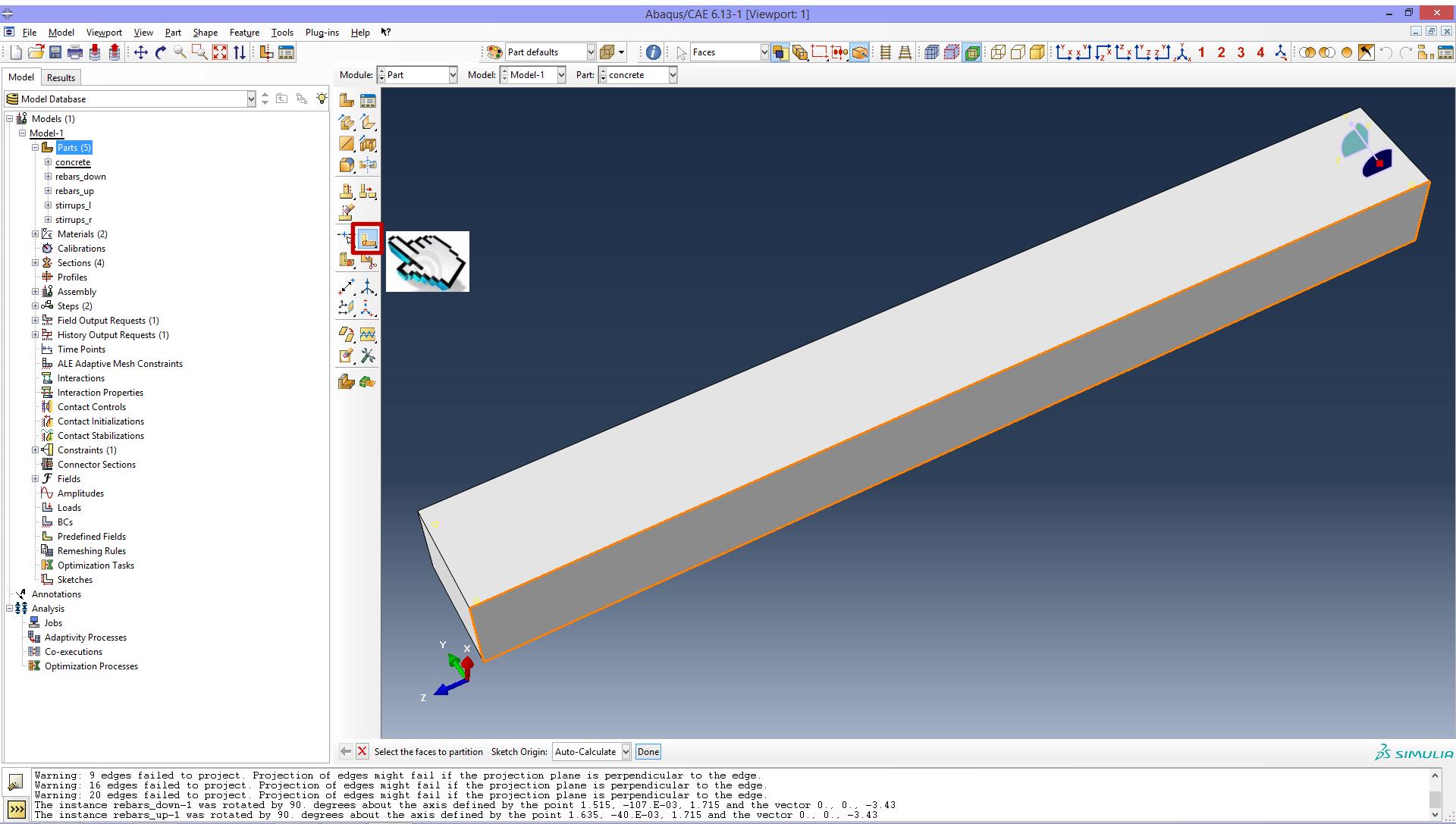
Model - Constraints

9. Connexions : connecter les barres d'acier à la poutre en béton



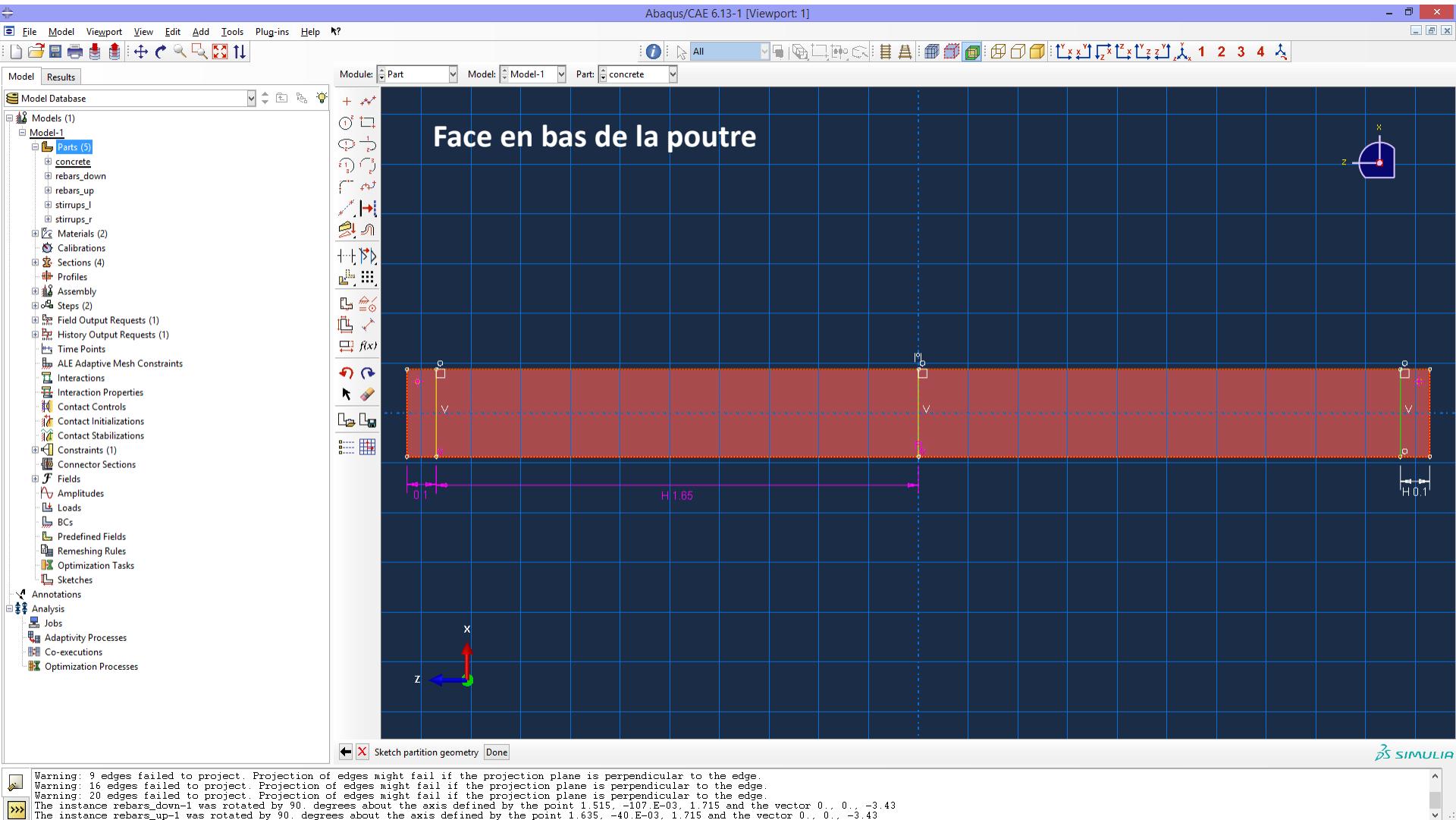
Model - Parts

10. Géométrie : dessin des parties où appliquer des conditions



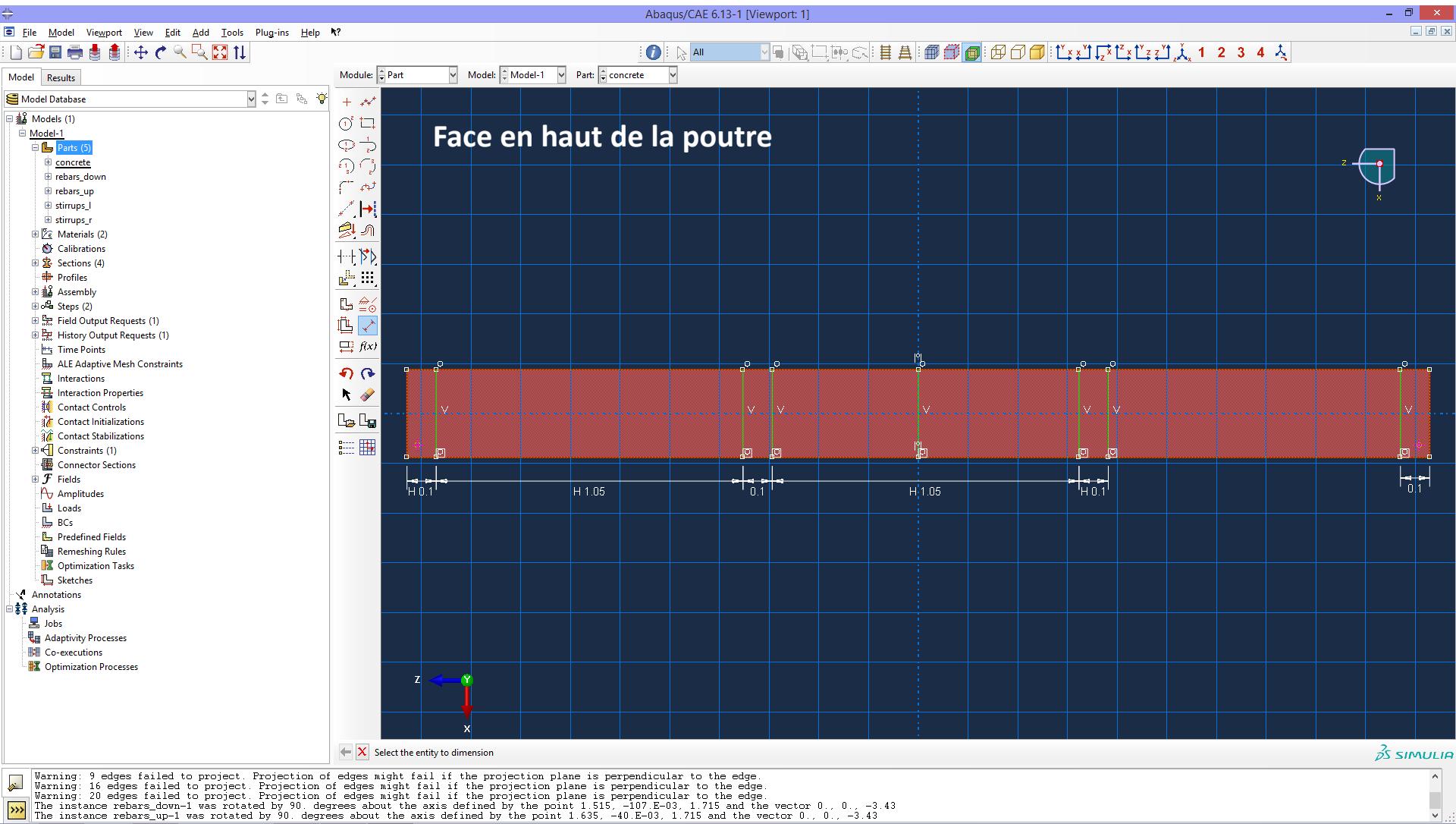
Model - Parts

10. Géométrie : dessin des parties où appliquer des conditions



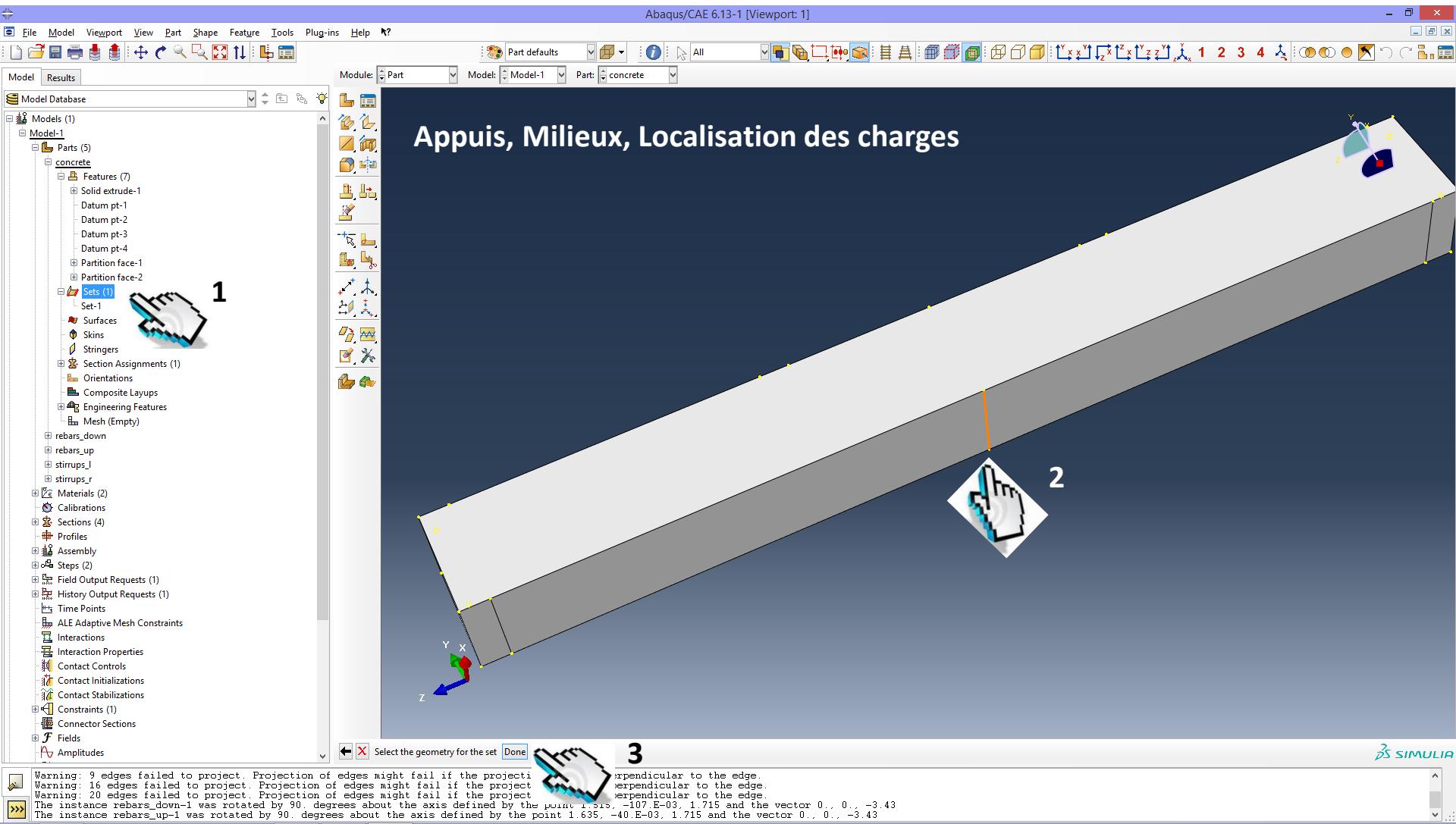
Model - Parts

10. Géométrie : dessin des parties où appliquer des conditions



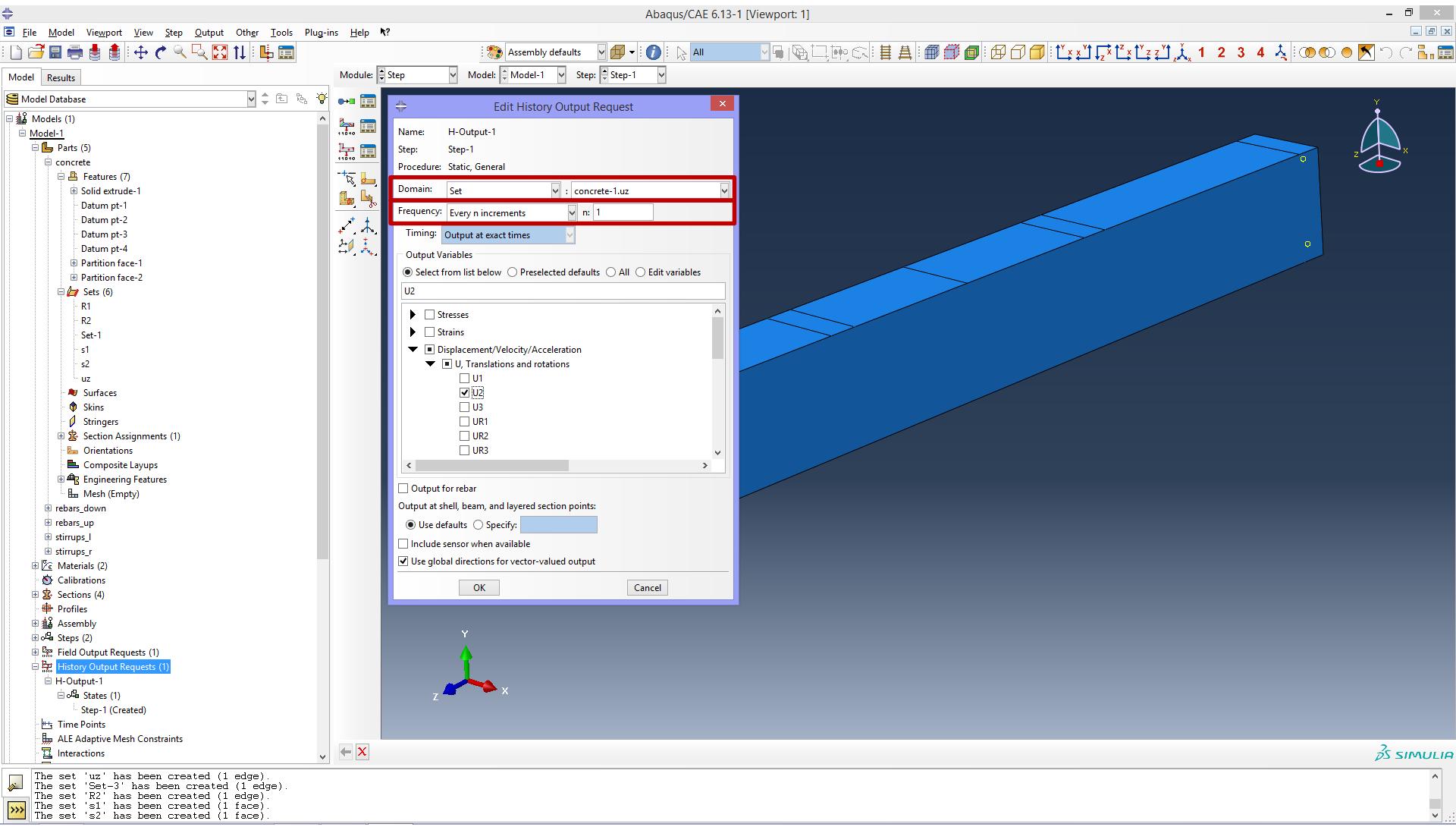
Model - Parts

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



Model - History Output Requests

12. Résultats dans le temps : réactions aux appuis, flèche, charges imposées



Model - BCs

13. Conditions aux limites : liaisons

Abaqus/CAE 6.13-1 [Viewport: 1]

File Model Viewport View Load BC Predefined Field Load Case Feature Tools Plugins Help ?

Module: Load Model: Model-1 Step: Step-1

Create Boundary Condition

Name: BC-1
Step: Step-1
Procedure: Static, General

Category: Mechanical
Fluid
Electrical/Magnetic
Other

Types for Selected Step:
Symmetry/Antisymmetry/Encastre
Displacement/Rotation
Velocity/Angular velocity
Connector displacement
Connector velocity

Continue... Cancel

Model Database

Models (1)
Model-1
Parts (5)
concrete
Features (7)
Sets (6)
Surfaces
Skins
Stringers
Section Assignments (1)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
rebars_down
rebars_up
stirrups_l
stirrups_r
Materials (2)
Calibrations
Sections (4)
Profiles
Assembly
Steps (2)
Field Output Requests (1)
History Output Requests (5)
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints (1)
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches

Annotations

Analysis

The set 'uz' has been created (1 edge).
The set 'Set-3' has been created (1 edge).
The set 'R2' has been created (1 edge).
The set 's1' has been created (1 face).
The set 's2' has been created (1 face).

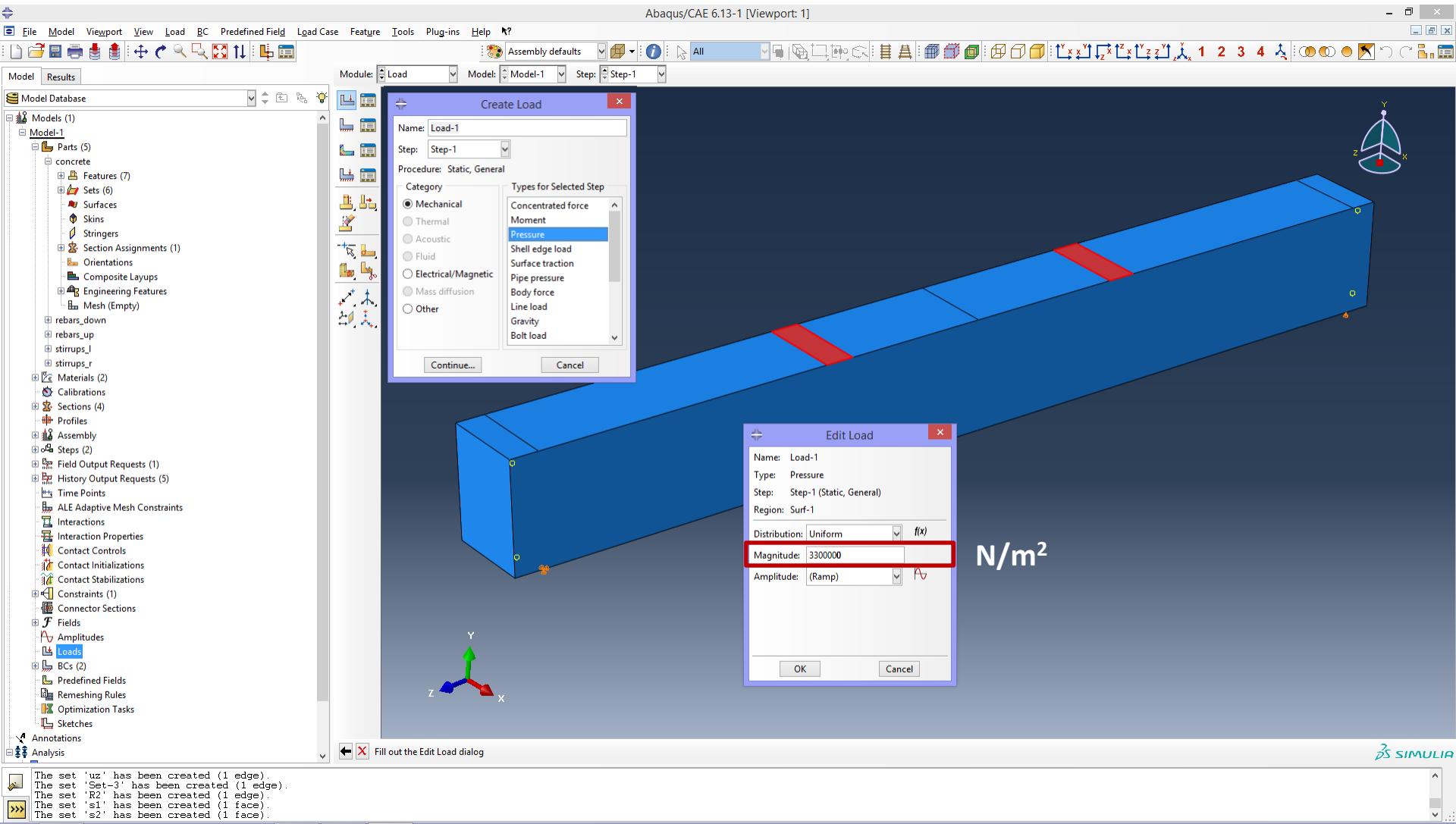
1. $u_y = 0, u_x = 0, u_z = 0$
2. $u_y = 0$

Y Z X

SIMULIA

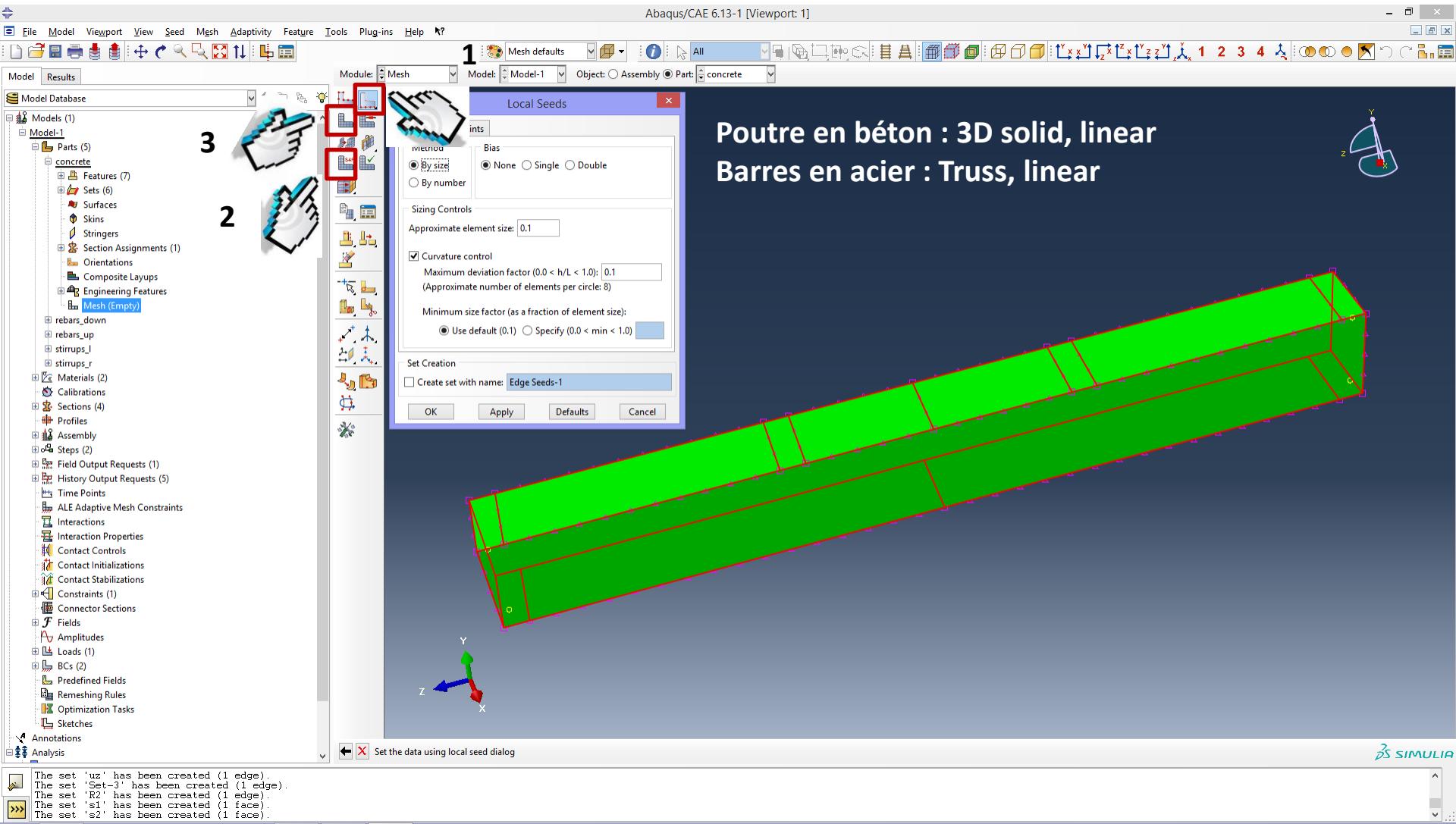
Model - Loads

14. Conditions aux limites : déplacement imposé



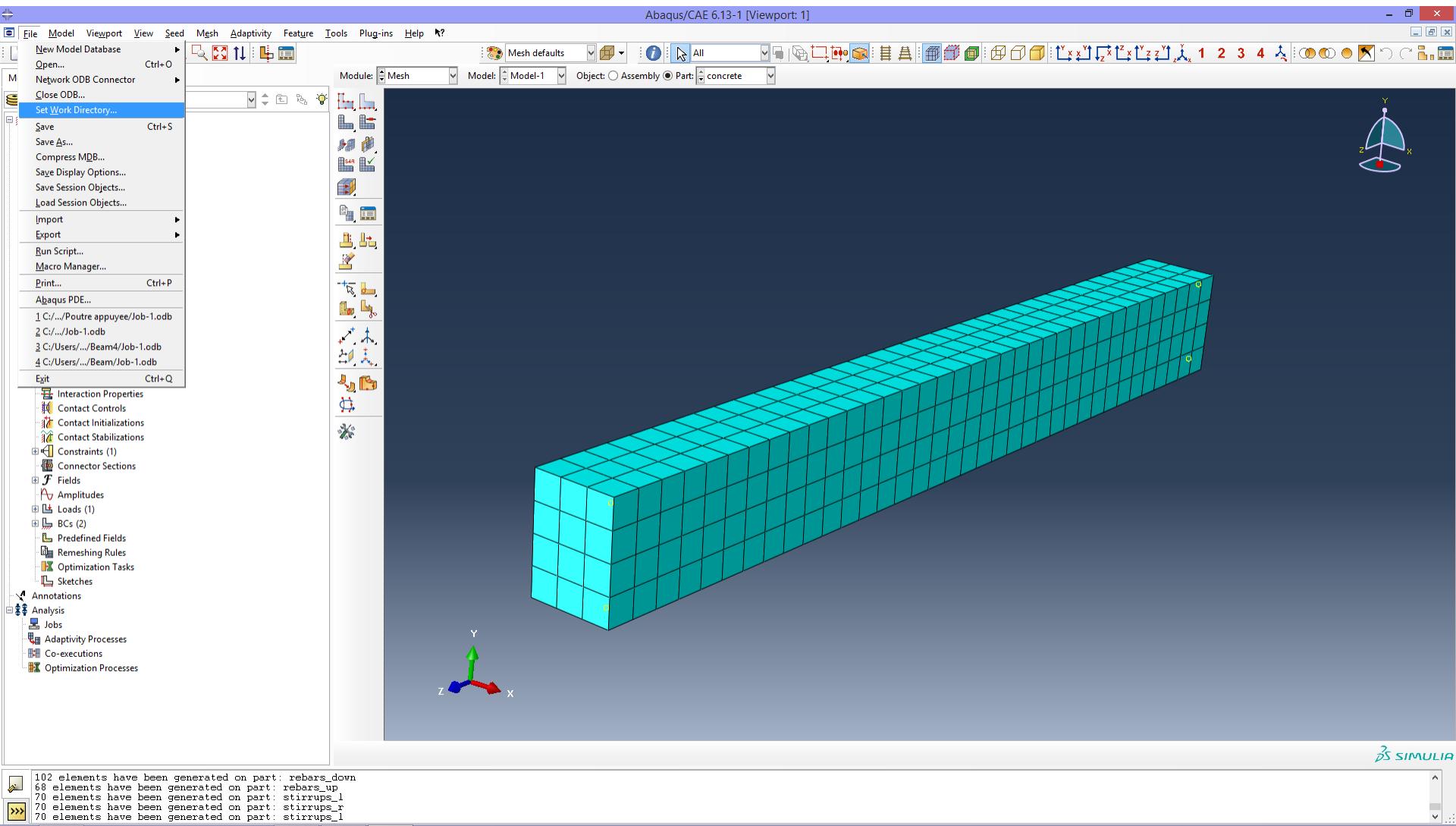
Model - Parts

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



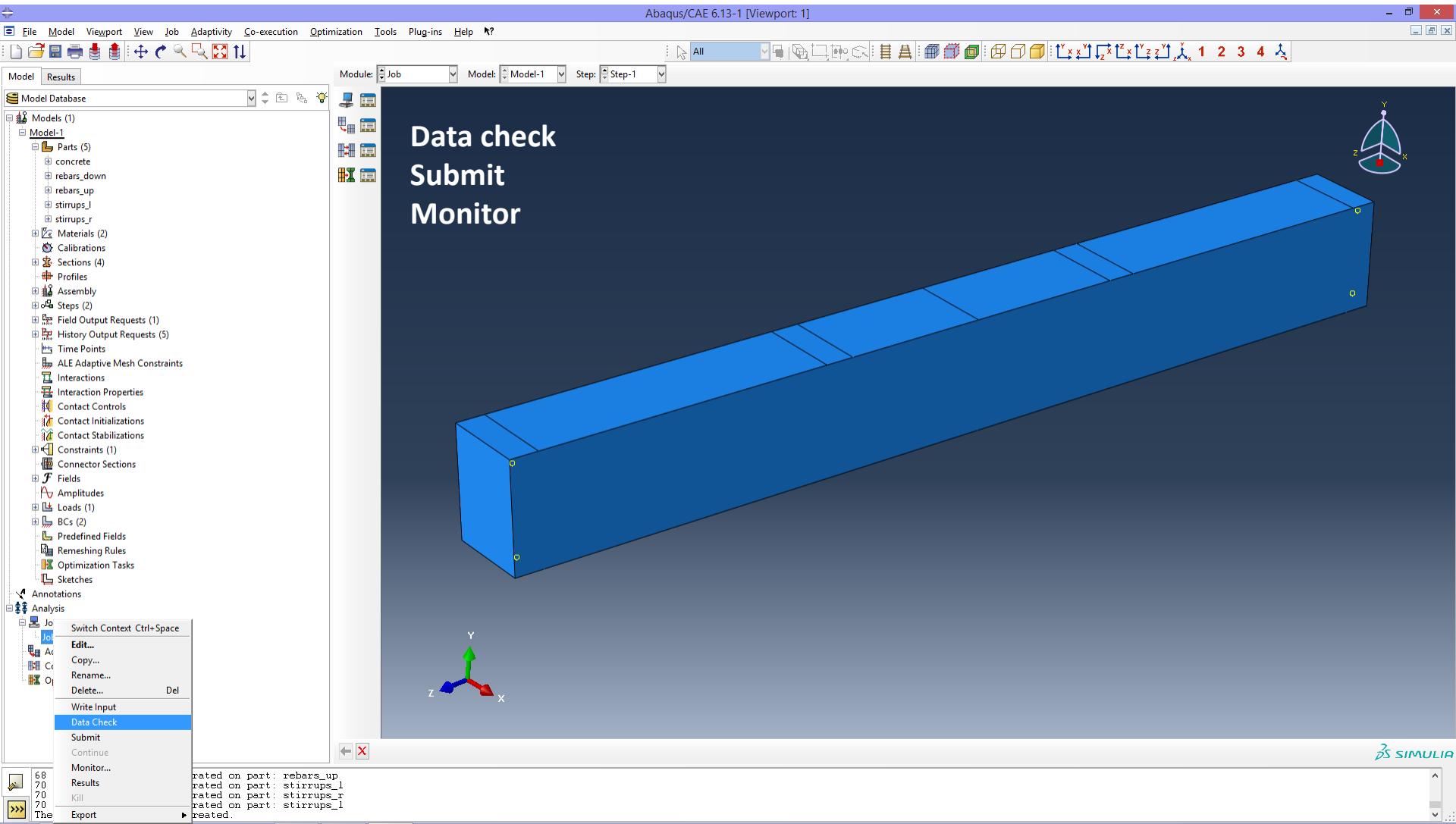
File - Save

19. Sélection du dossier et sauvegarde

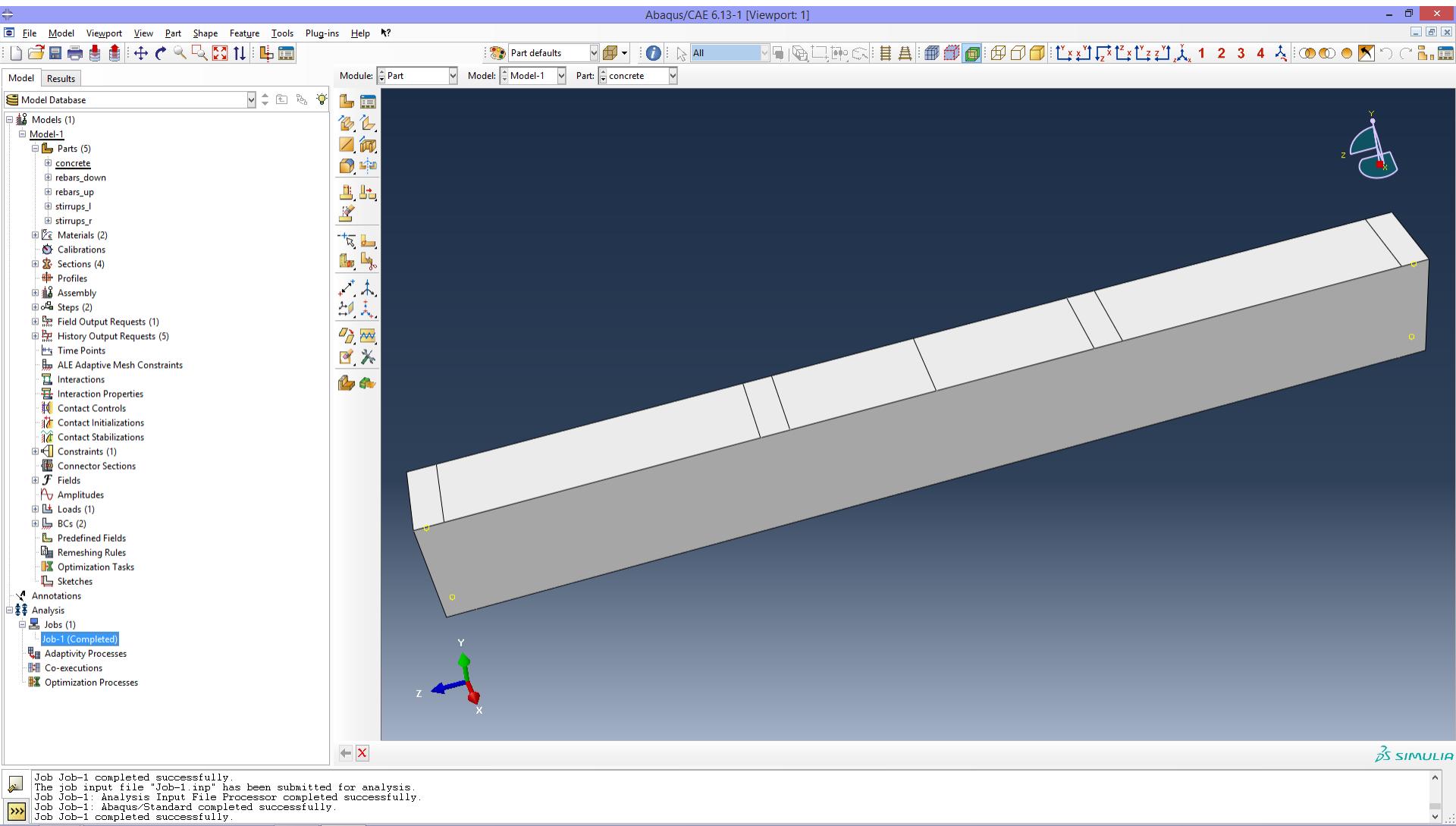


Model - Analysis

20. Création du projet, vérification des données et calcul

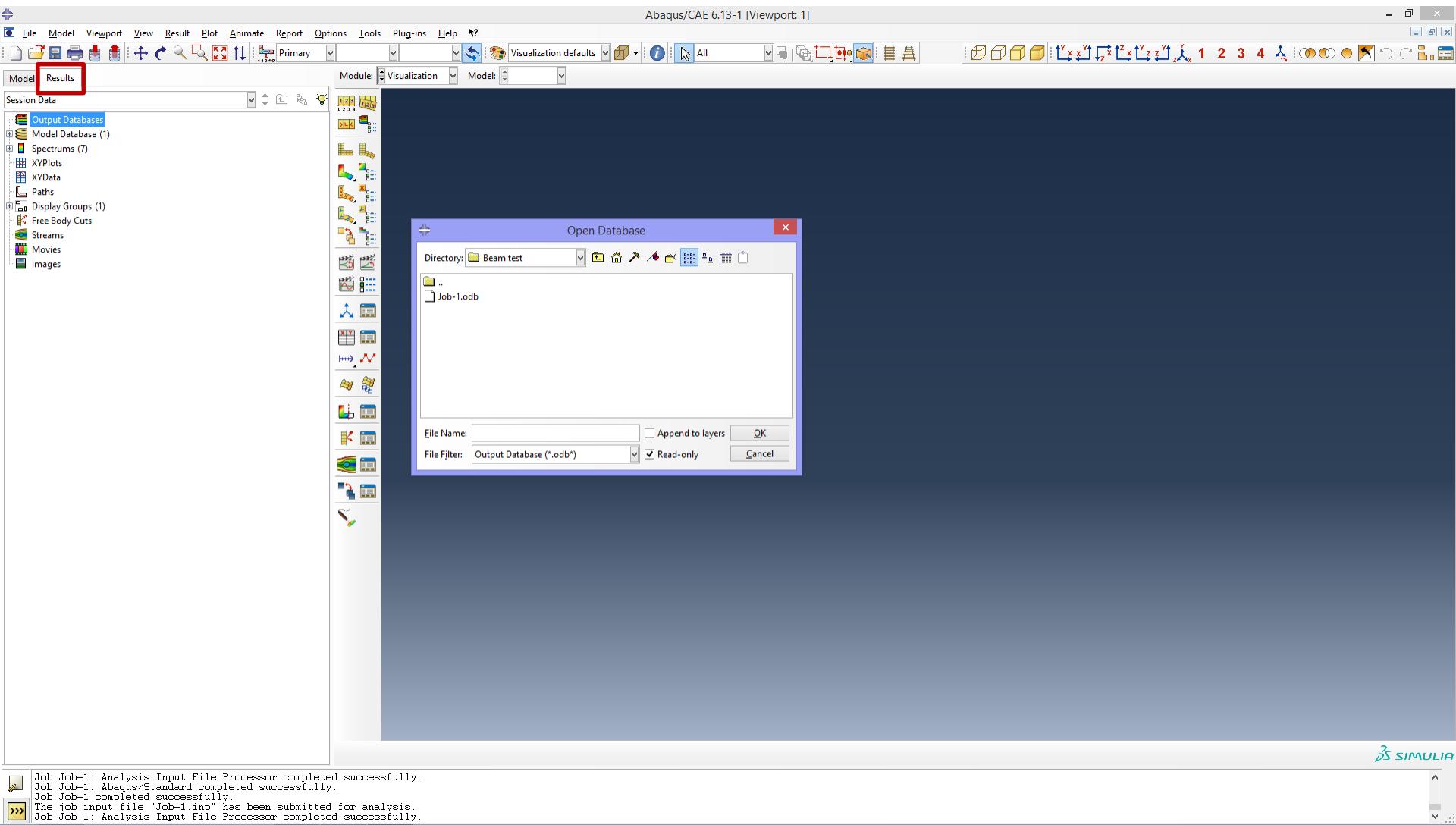


Calcul complet



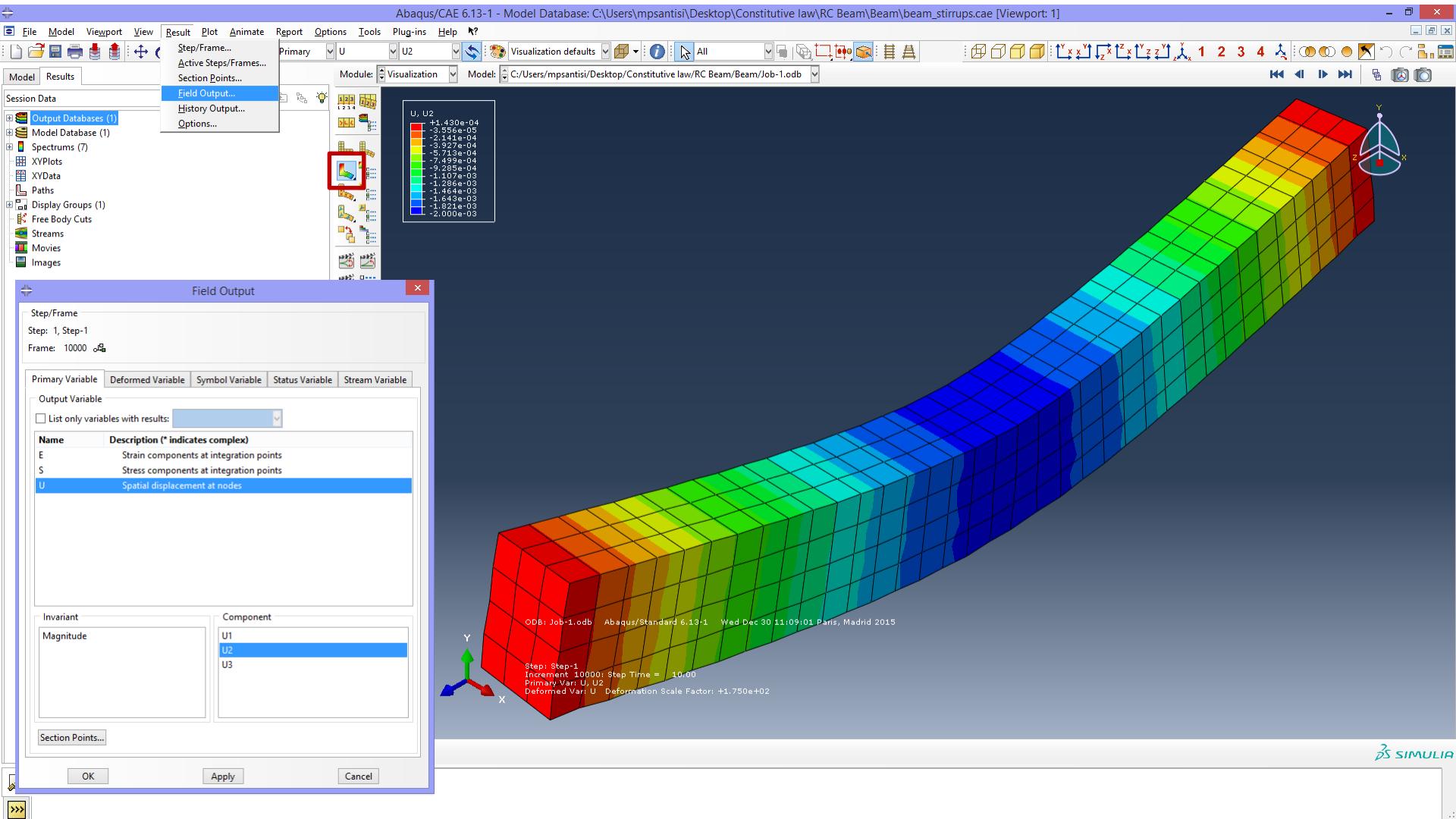
Results

1. Fichier de résultats : *.odb



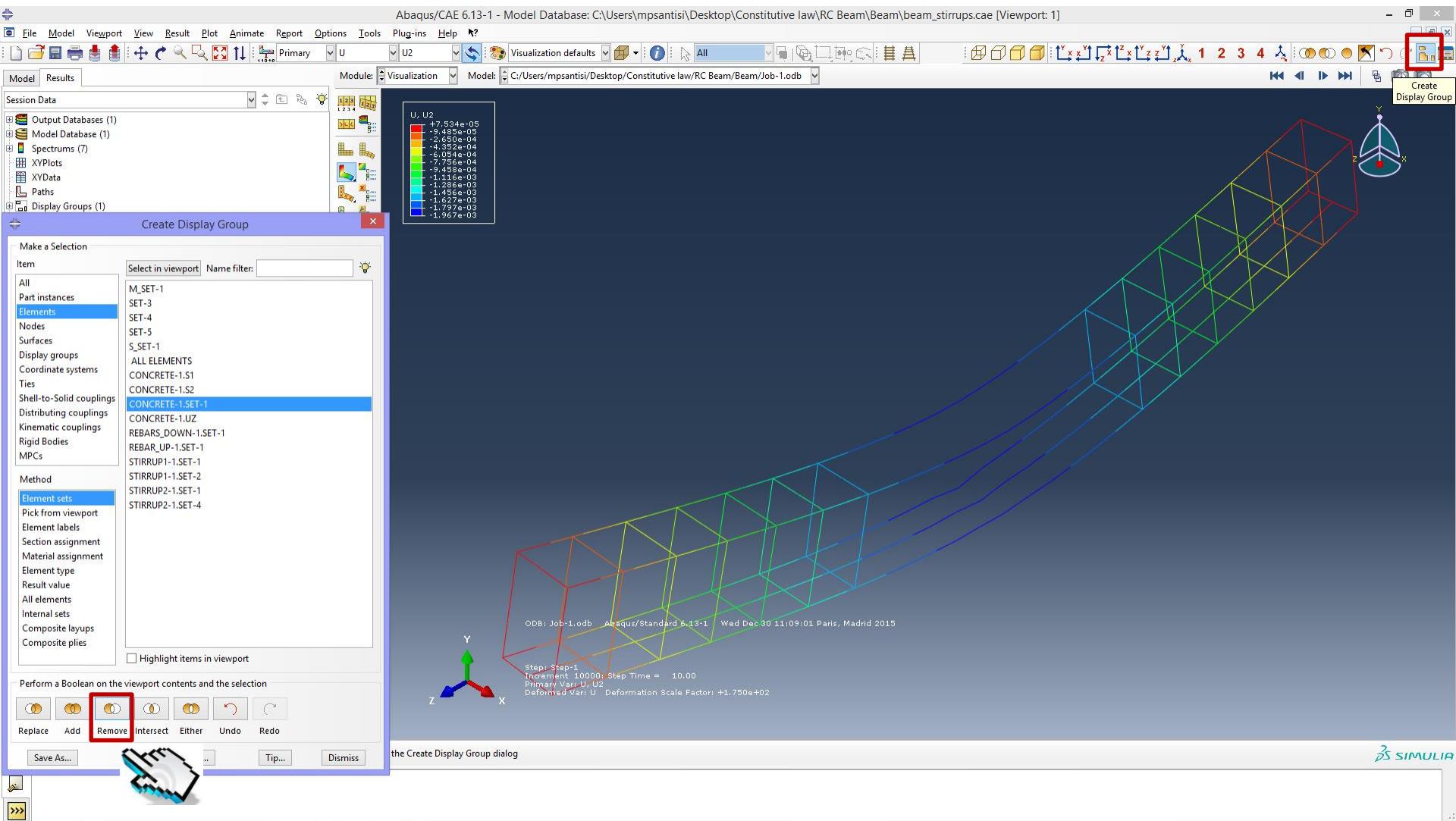
Results

2. Déformée et cartographie : déformation u2



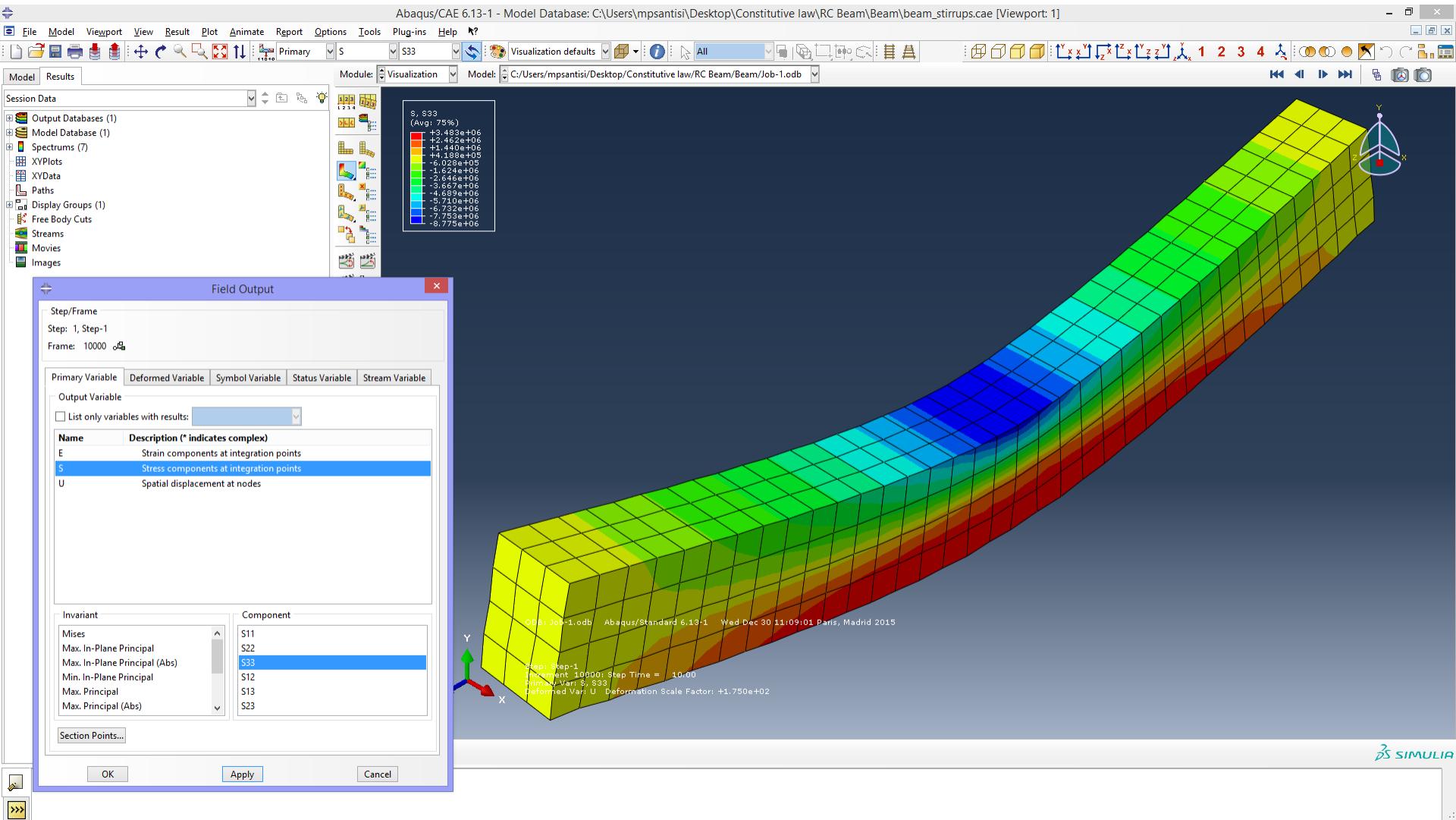
Results

2. Déformée et cartographie : déformation u2 des barres



Results

2. Déformée et cartographie : contrainte s33



Results

3. Contraintes dans le cas de béton à comportement élasto-plastique asymétrique

