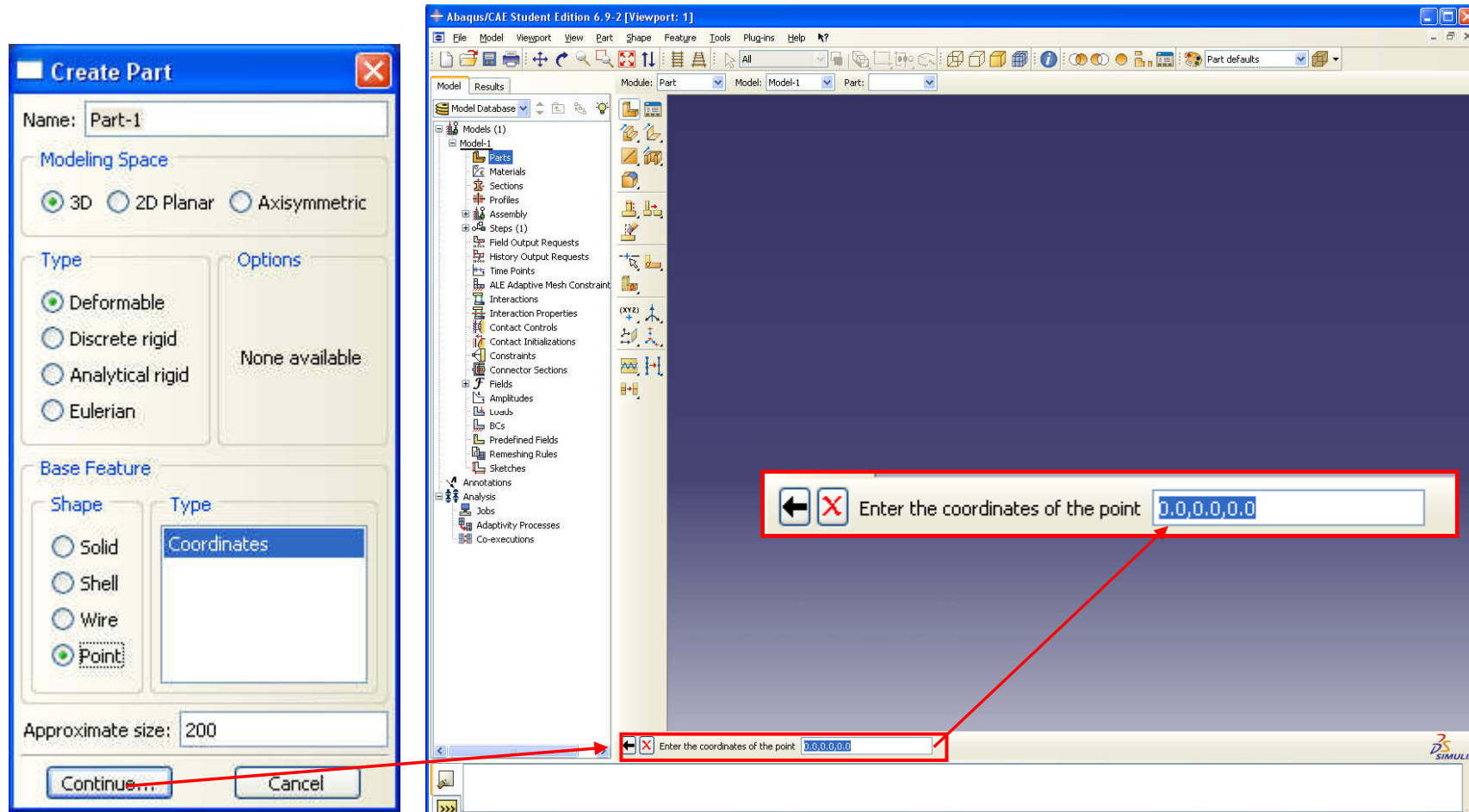
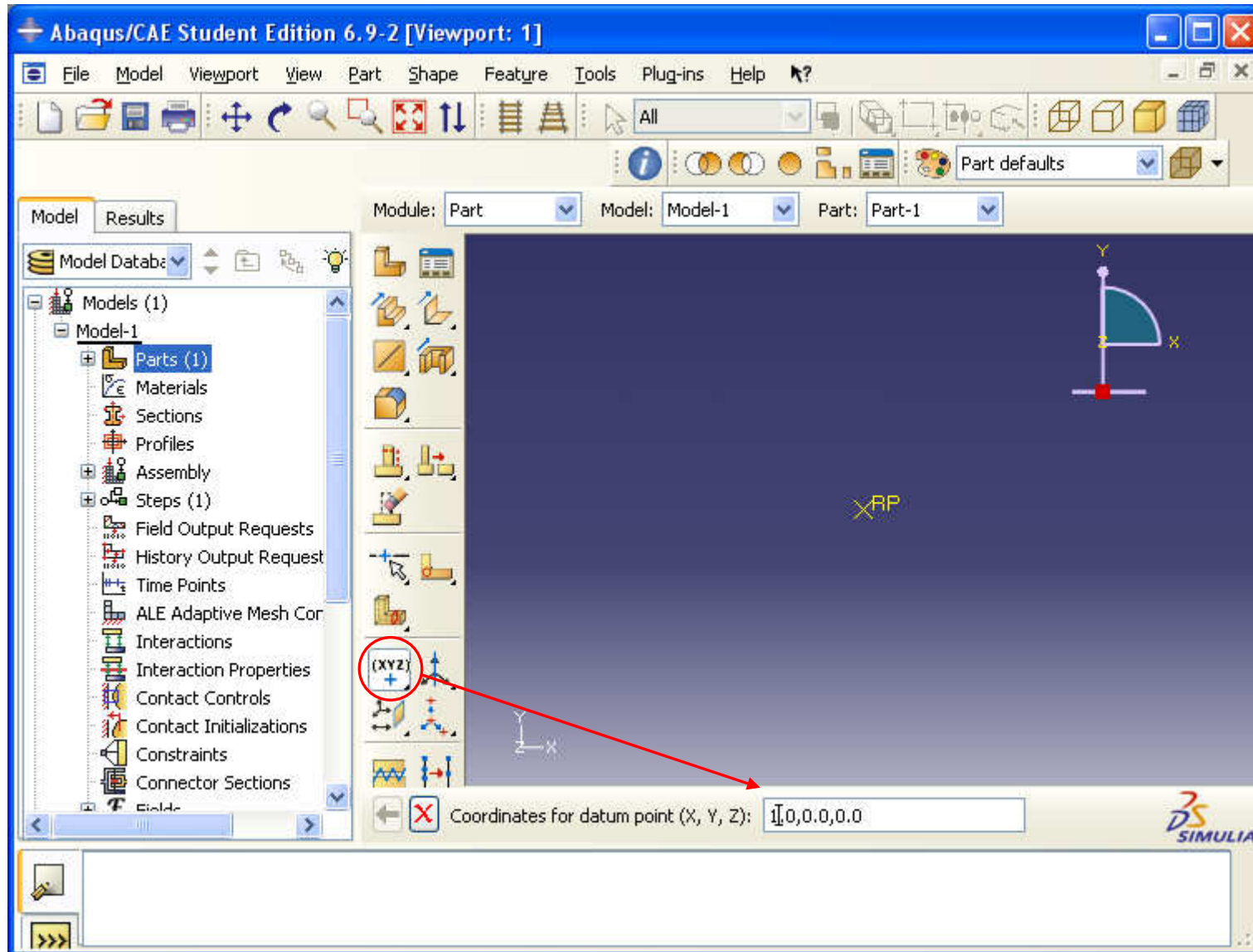


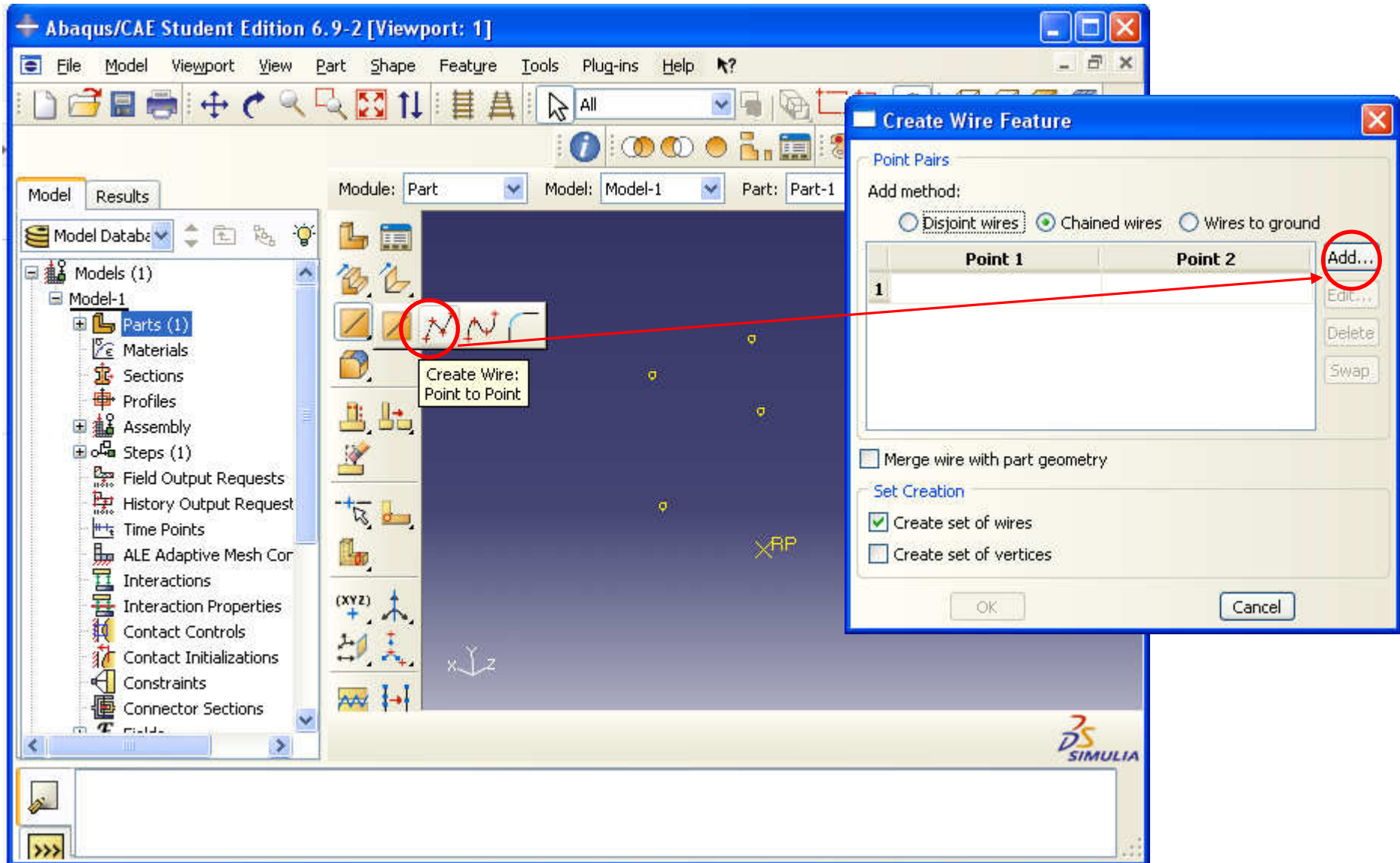
# ABAQUS Beams



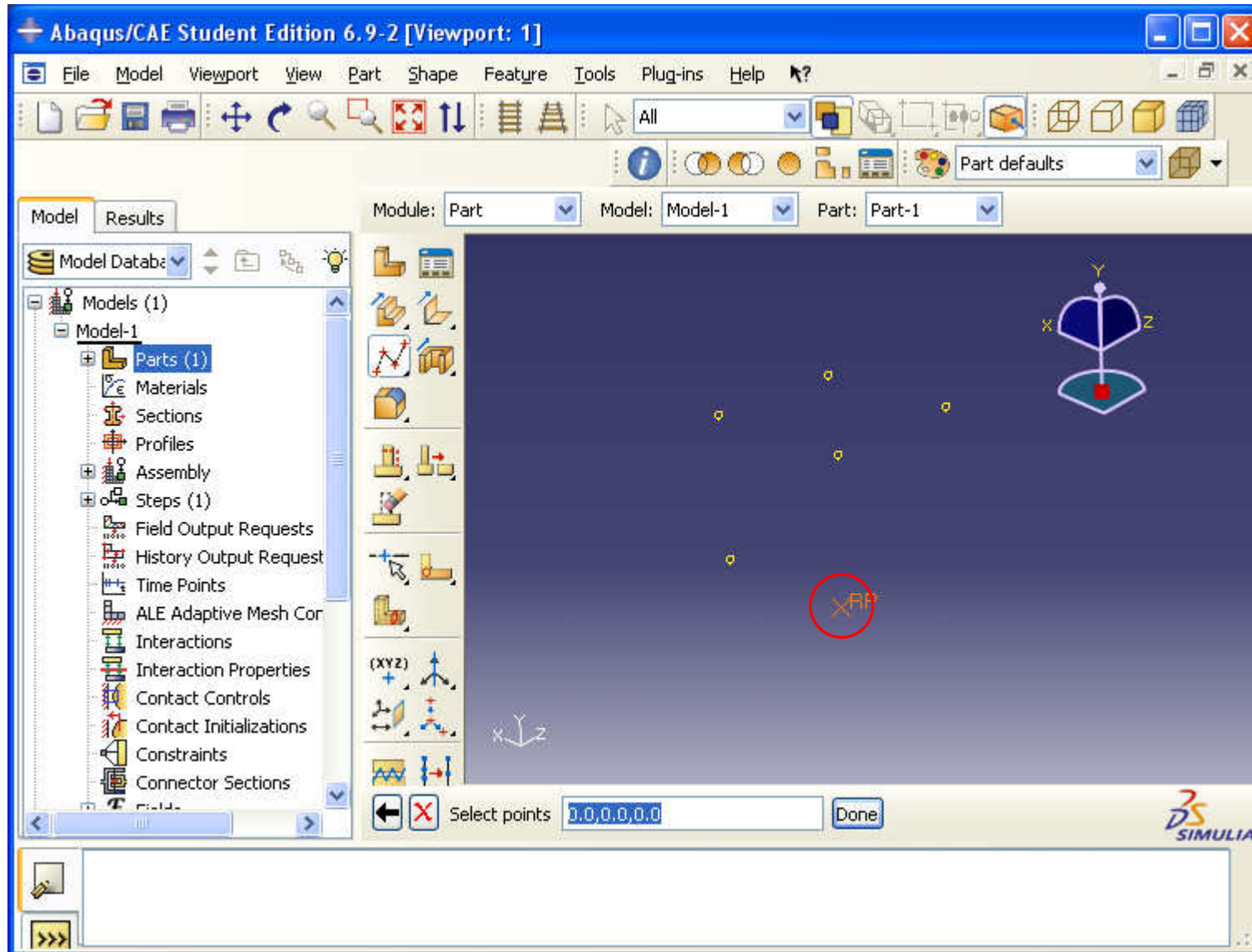
# Geometry nodes



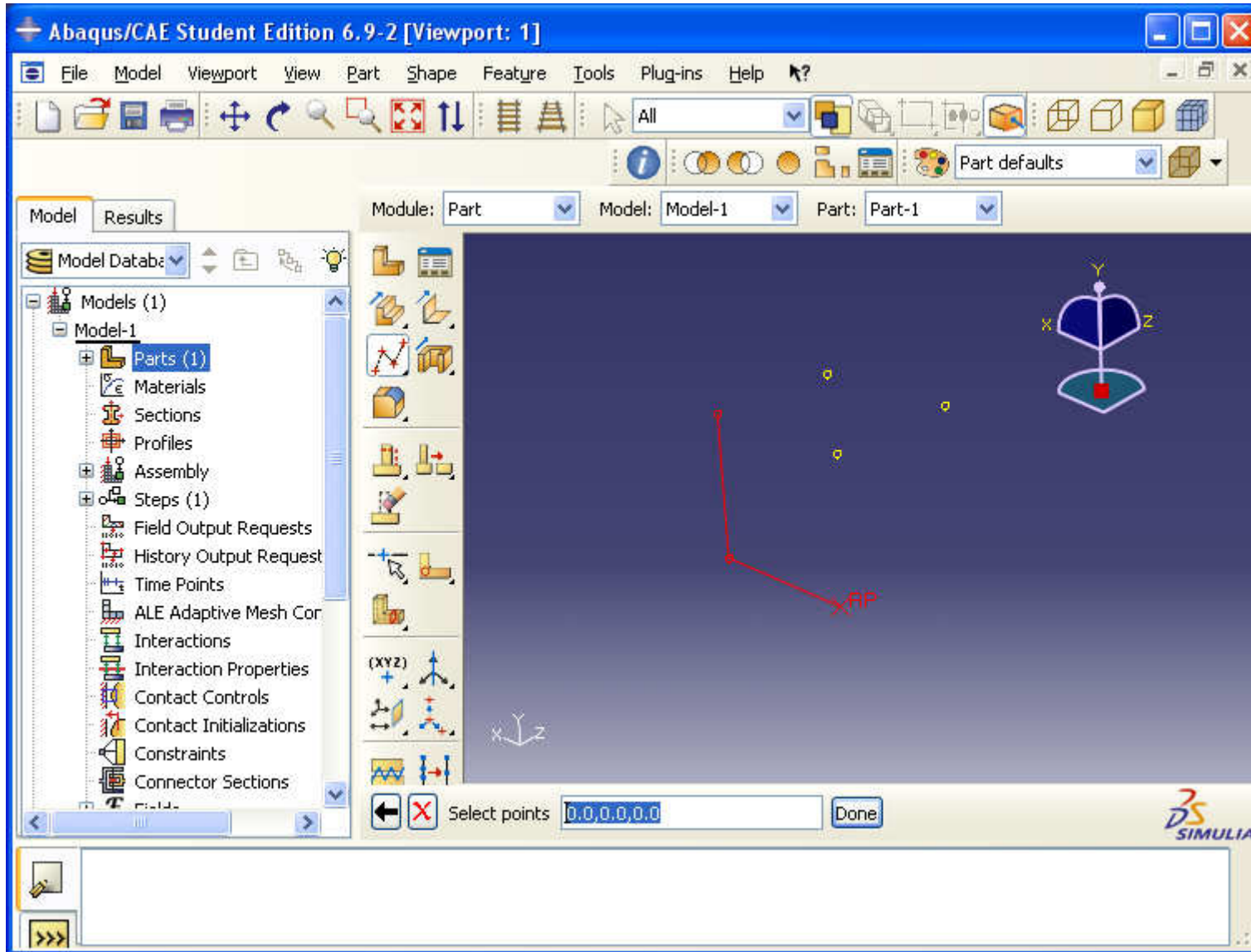
# Create wire



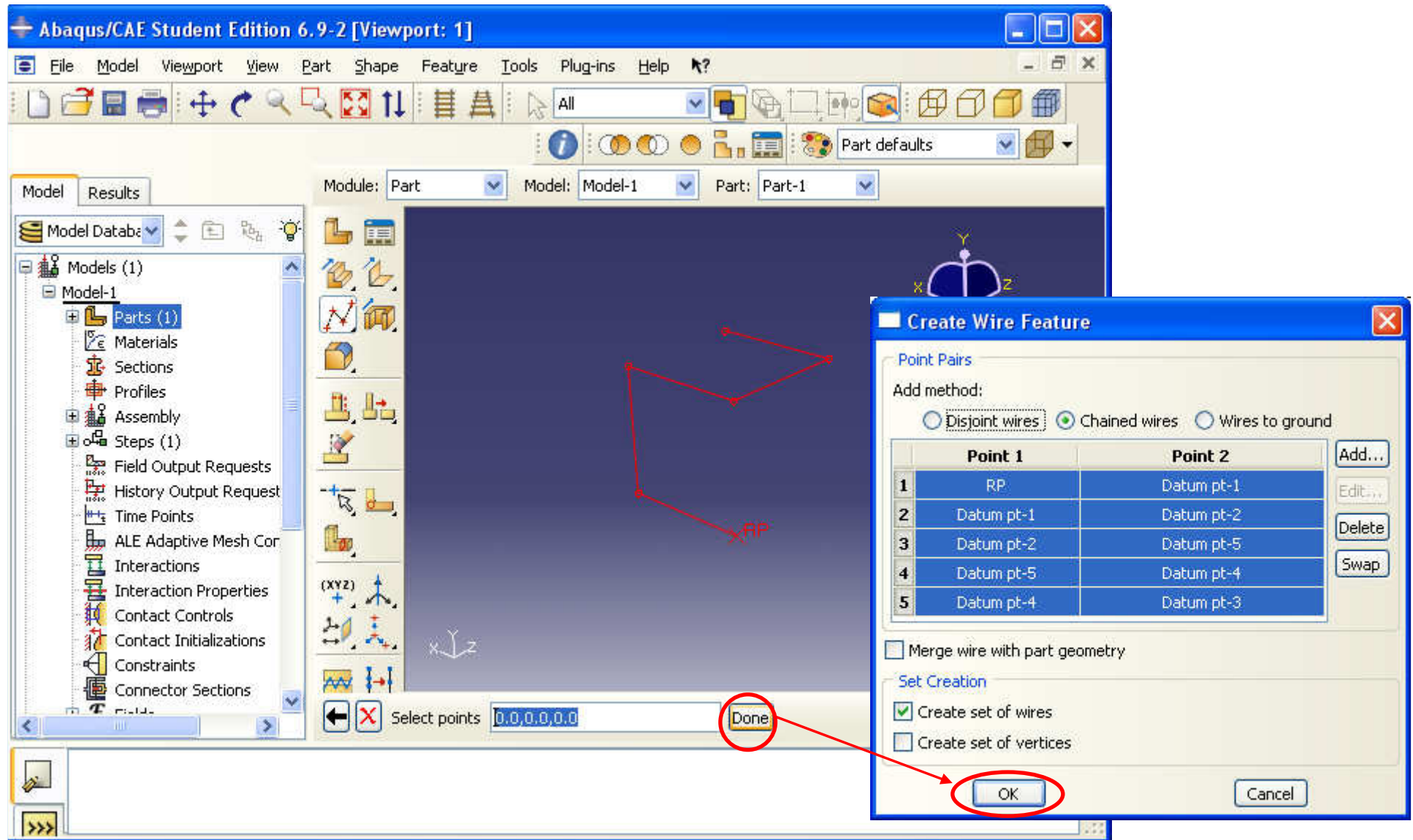
# First point



# Create wire



# Create wire



$E=200\text{GPa}$

$\nu=0.3$

# Materials data

The image shows a screenshot of the Abaqus/CAE Student Edition 6.9-2 software interface. The main window displays a 3D model of a part with a coordinate system (X, Y, Z). The left-hand side contains a tree view with the following structure:

- Model Database
- Models (1)
  - Model-1
    - Parts (1)
    - Materials (1)
    - Sections
    - Profiles
    - Assembly
    - Steps (1)
    - Field Output Requests
    - History Output Request
    - Time Points
    - ALE Adaptive Mesh Cor
    - Interactions
    - Interaction Properties
    - Contact Controls
    - Contact Initializations
    - Constraints
    - Connector Sections

**Create Profile**

Name: Profile-1

Shape

- Box
- Pipe
- Circular
- Rectangular**
- Hexagonal
- Trapezoidal
- I
- L
- T
- Arbitrary
- Generalized

Continue... Cancel

**Edit Profile**

Name: Profile-1

Shape: Rectangular

a: 0.05

b: 0.1

OK Cancel

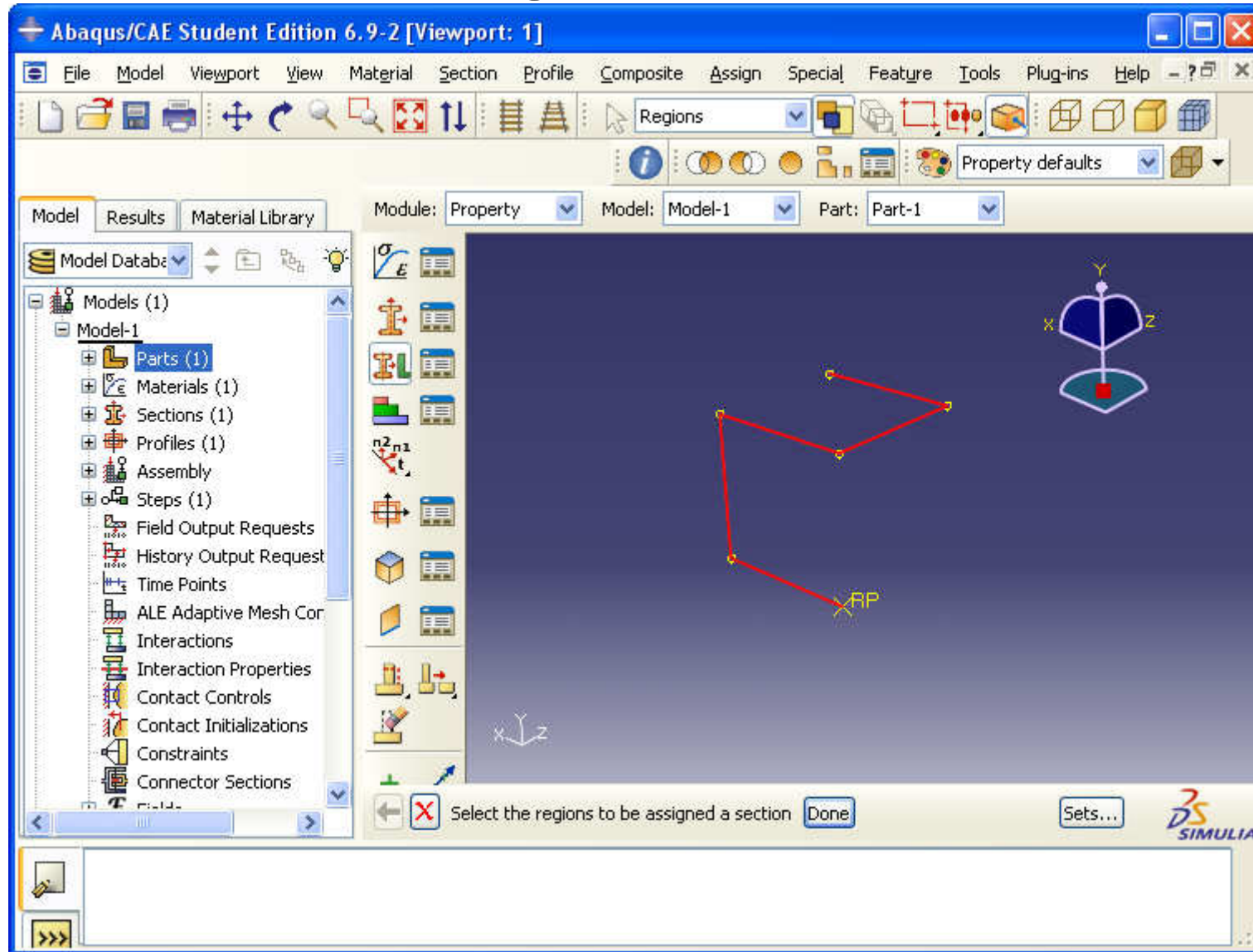
# Create section

The image displays the Abaqus/CAE Student Edition 6.9-2 interface. The main window shows a 3D model of a structure with a red circle highlighting the 'Create Section' icon in the toolbar. Two dialog boxes are overlaid on the interface:

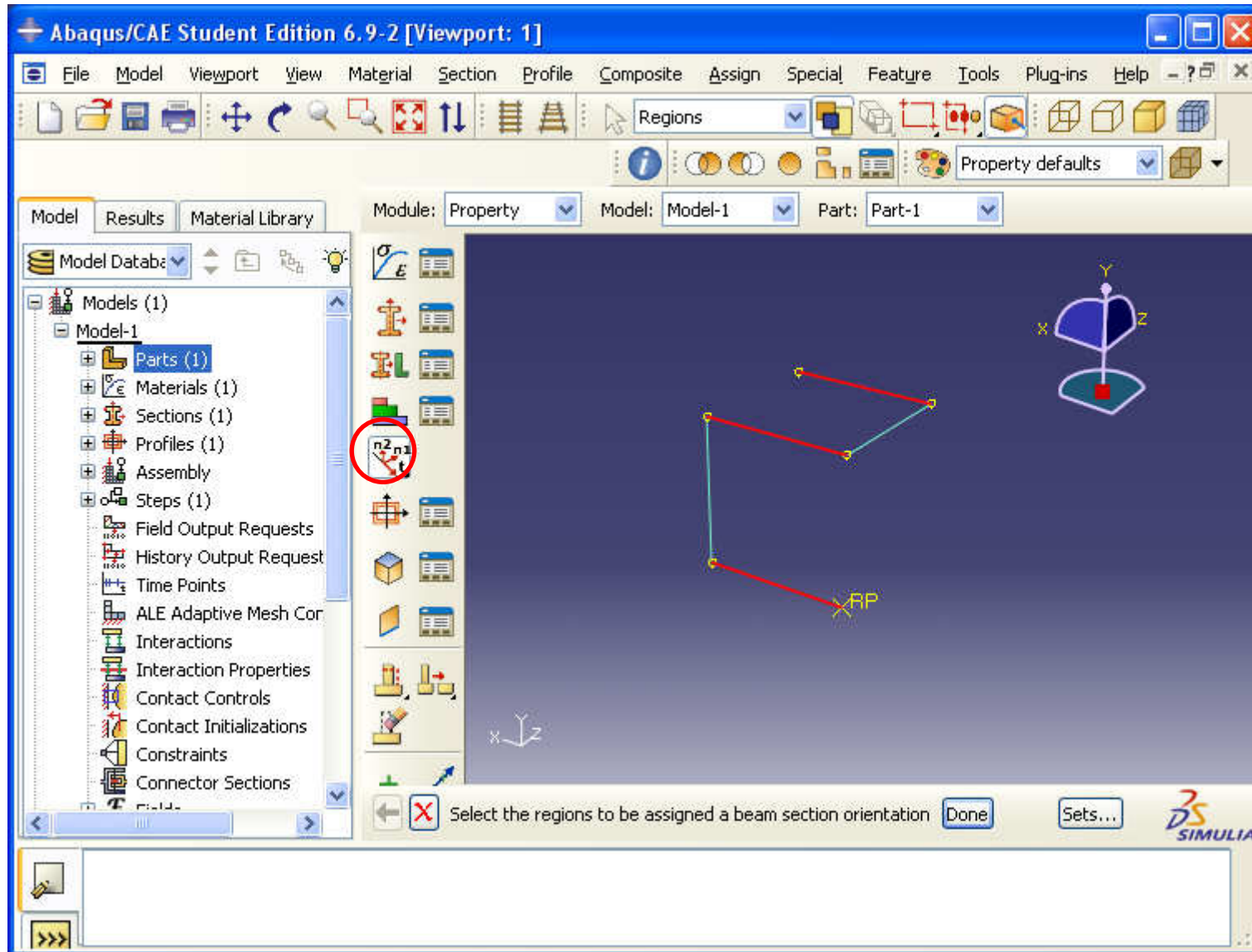
- Create Section Dialog:**
  - Name: Section-1
  - Category:  Solid,  Shell,  Beam,  Other
  - Type: Beam (selected), Truss
  - Buttons: Continue..., Cancel
- Edit Beam Section Dialog:**
  - Name: Section-1
  - Type: Beam
  - Section integration:  During analysis,  Before analysis
  - Profile name: Profile-1 (selected), Create...
  - Profile shape: Basic (selected), Stiffness, Fluid Inertia
  - Material name: Material-1, Create...
  - Section Poisson's ratio: 0.3
  - Temperature variation:  Linear by gradients,  Interpolated from temperature points
  - Buttons: OK, Cancel



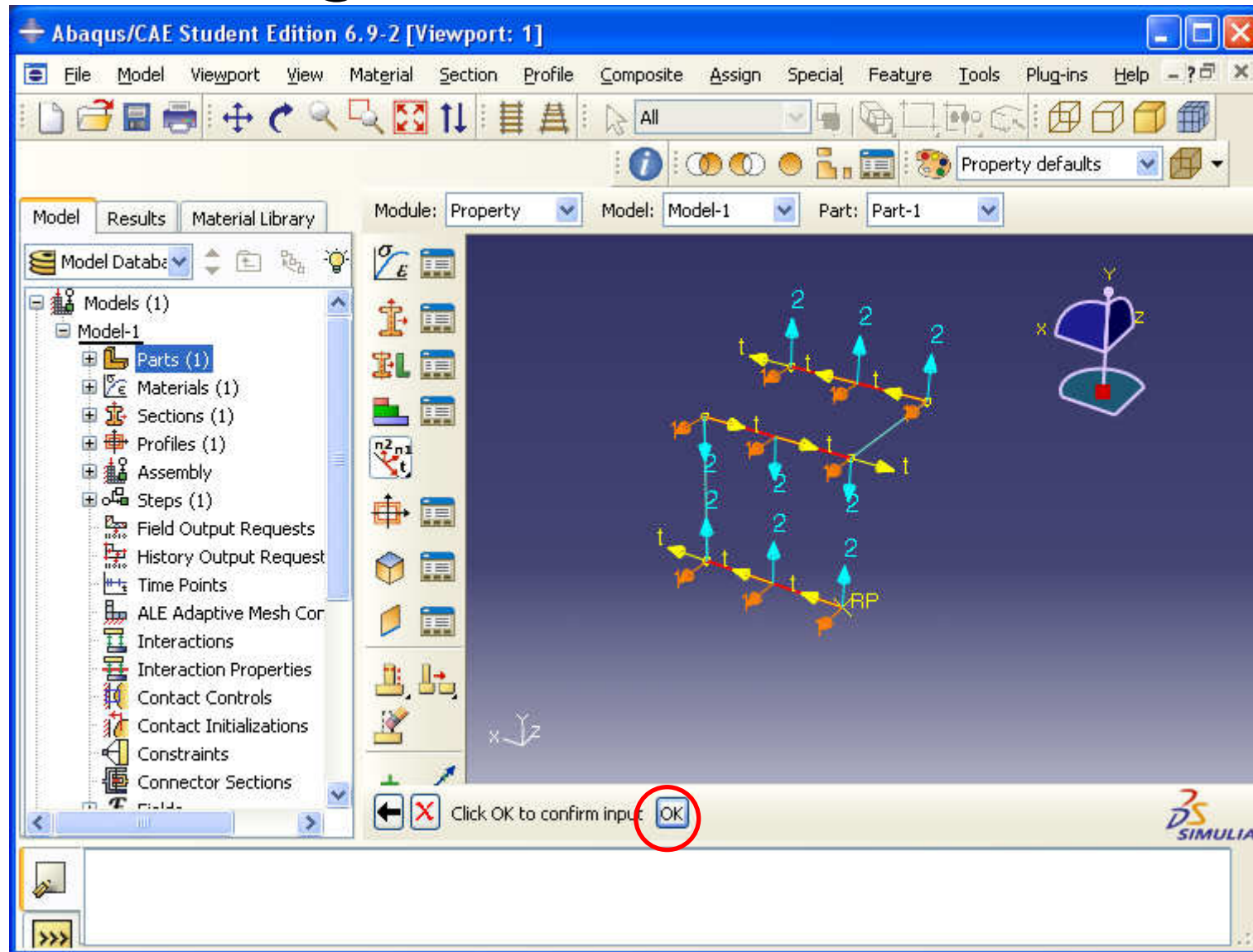
# Assign section



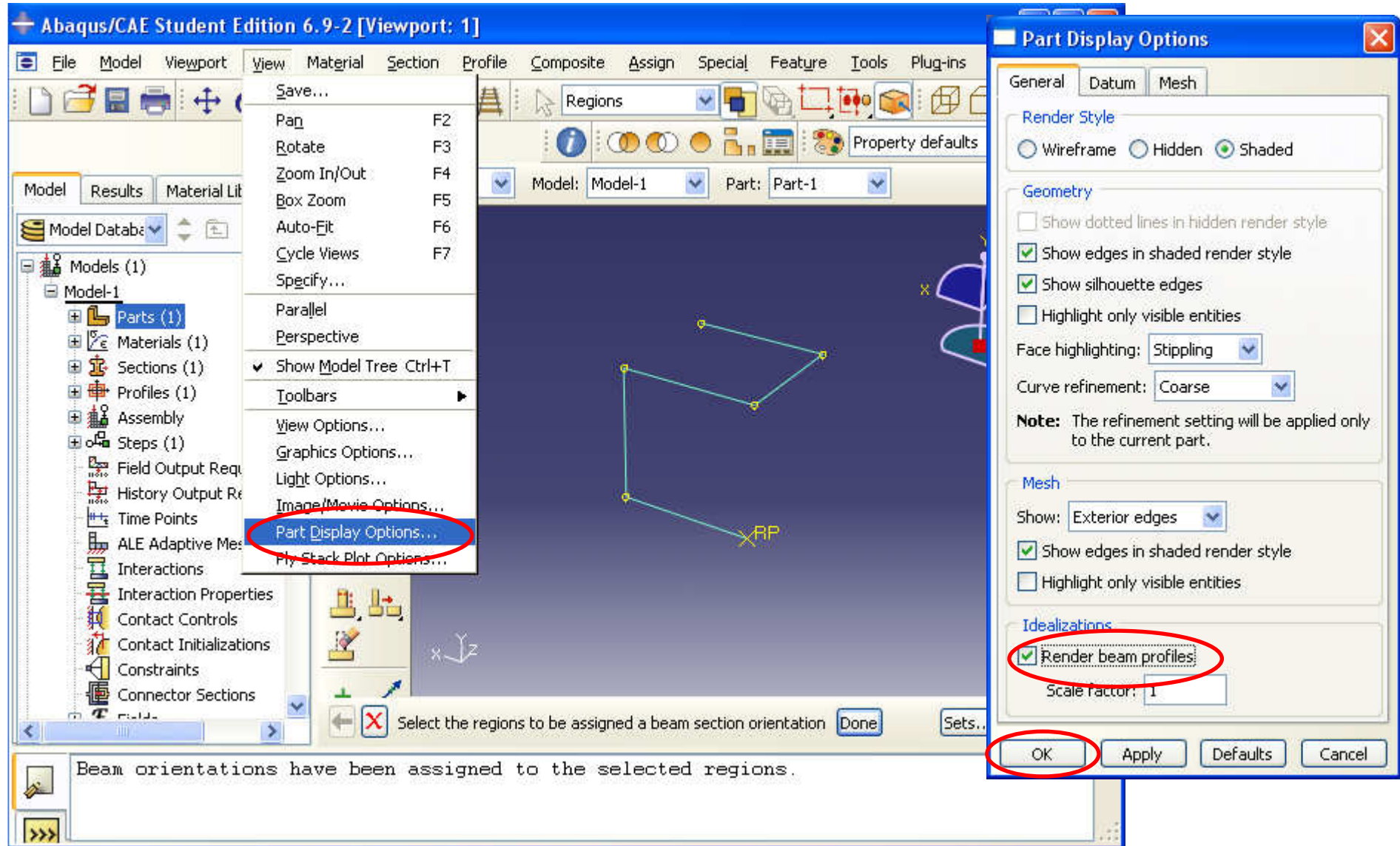
# Assign beam orientation



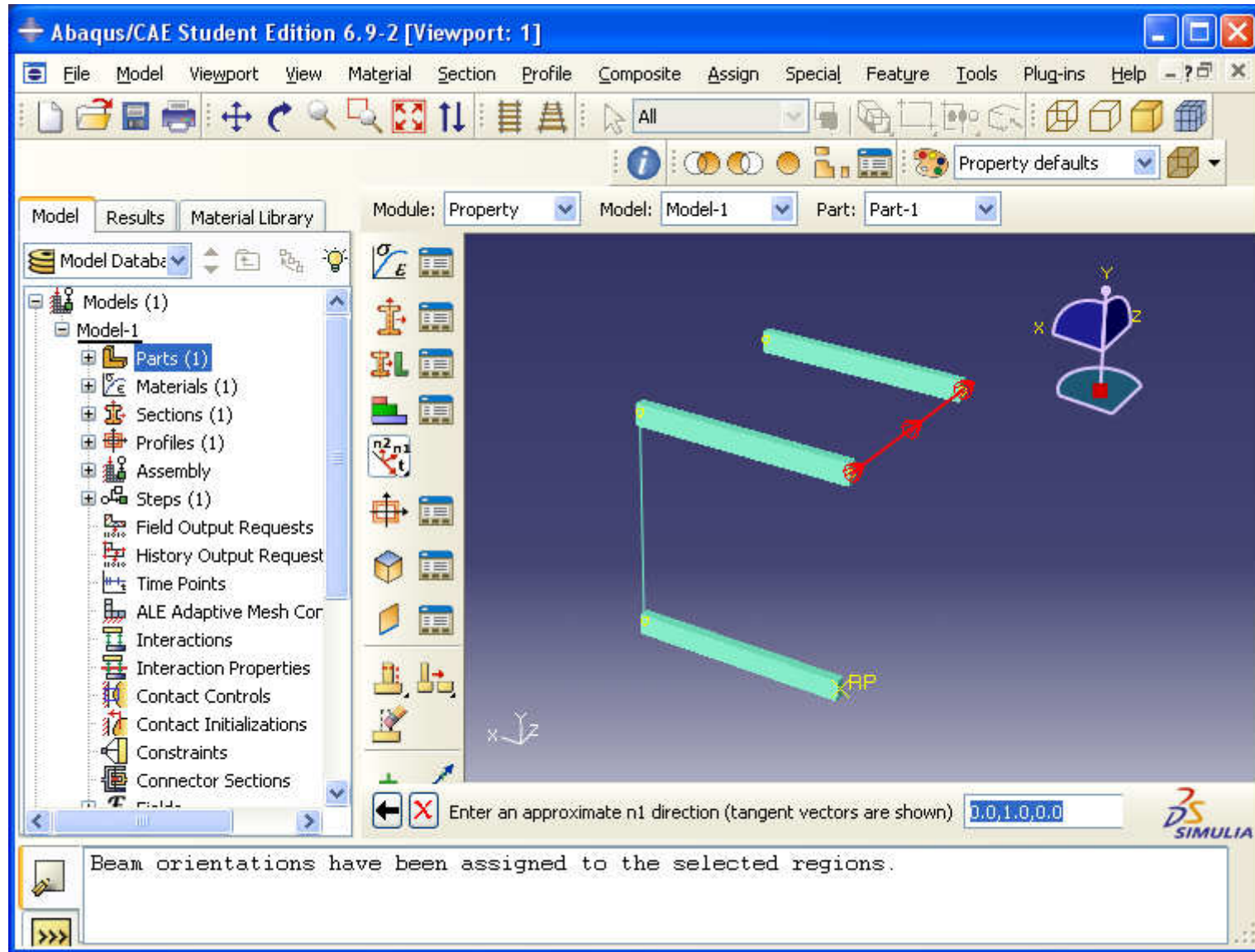
# Assign beam orientation



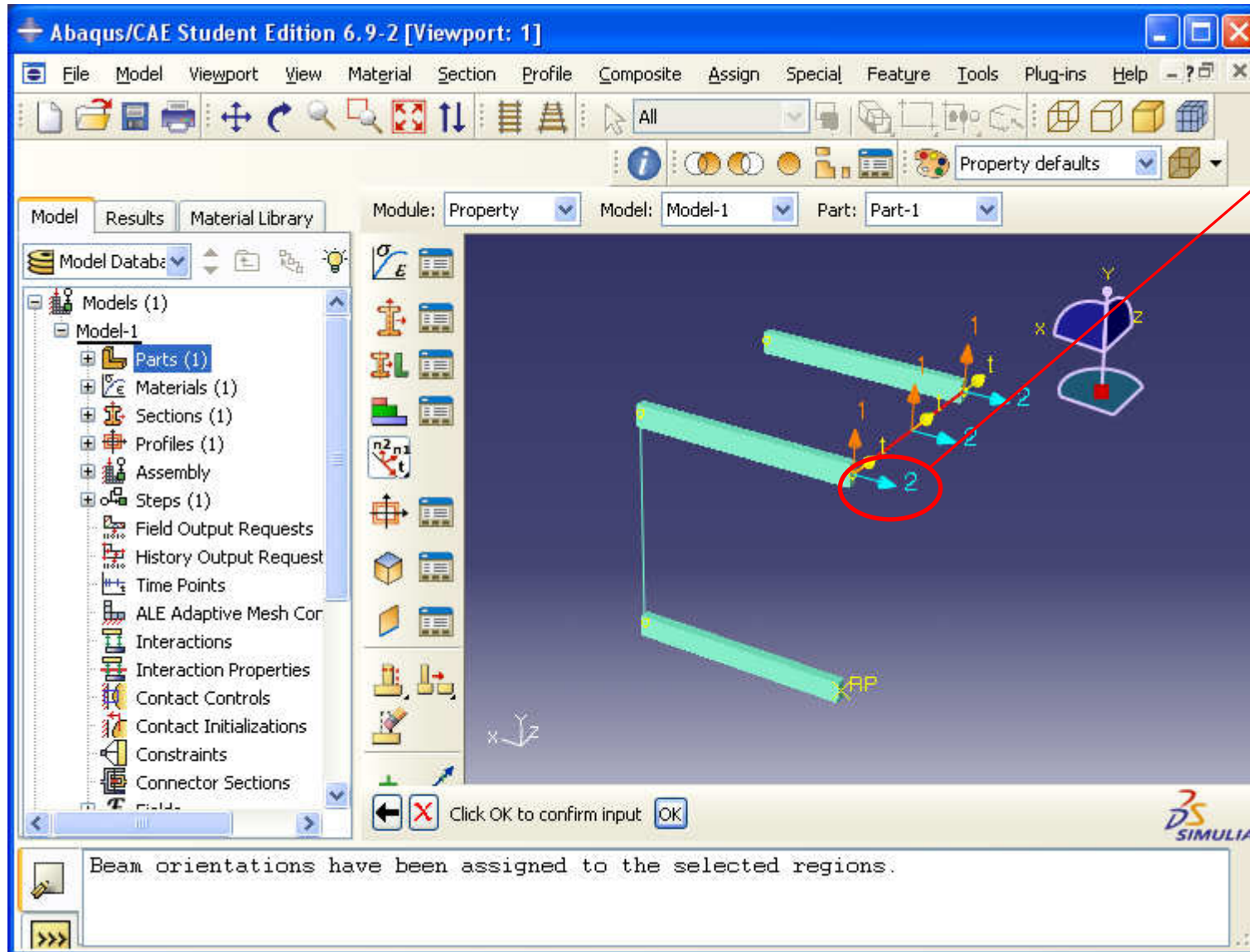
# Render beam profile



# Assign beam orientation

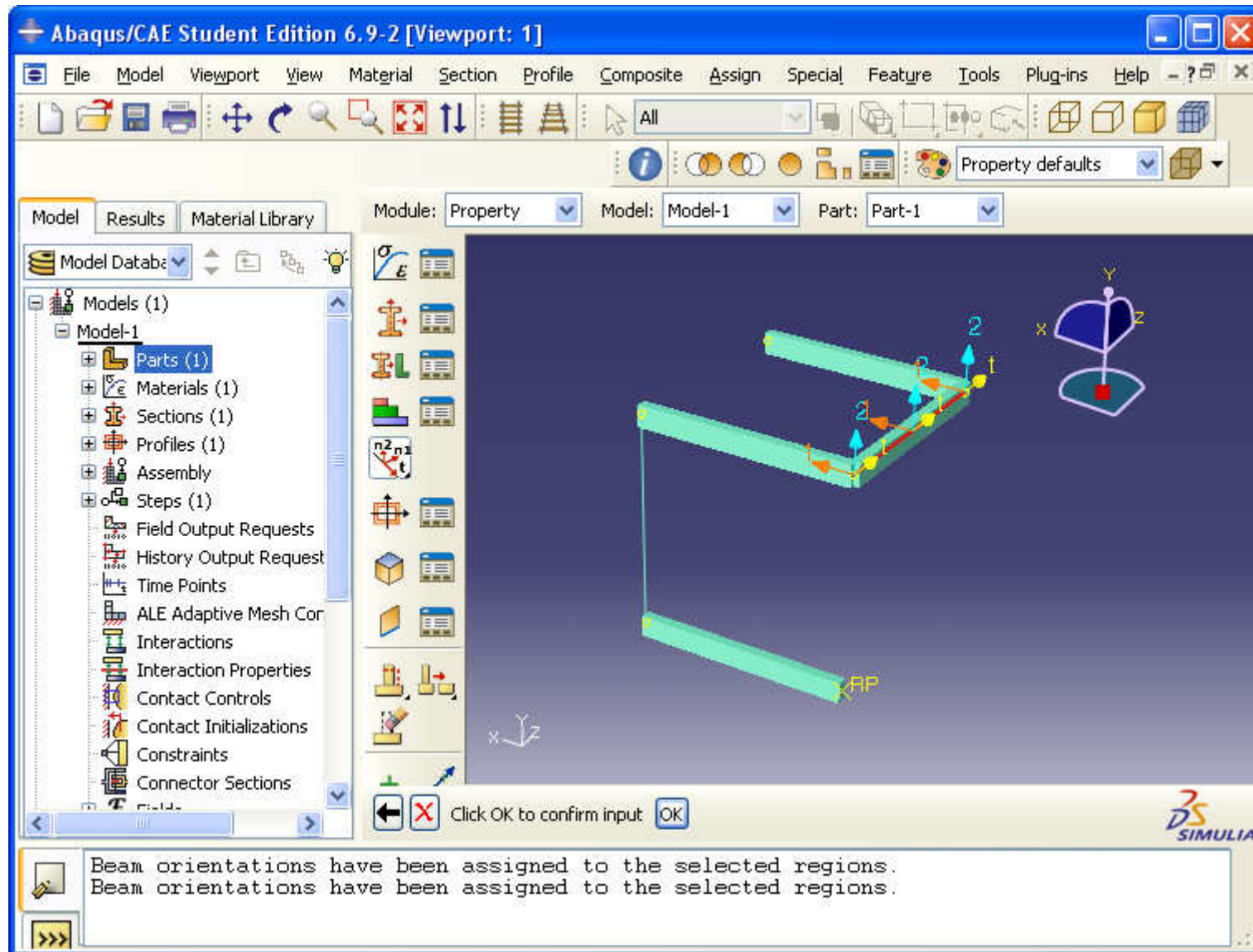


# Assign beam orientation

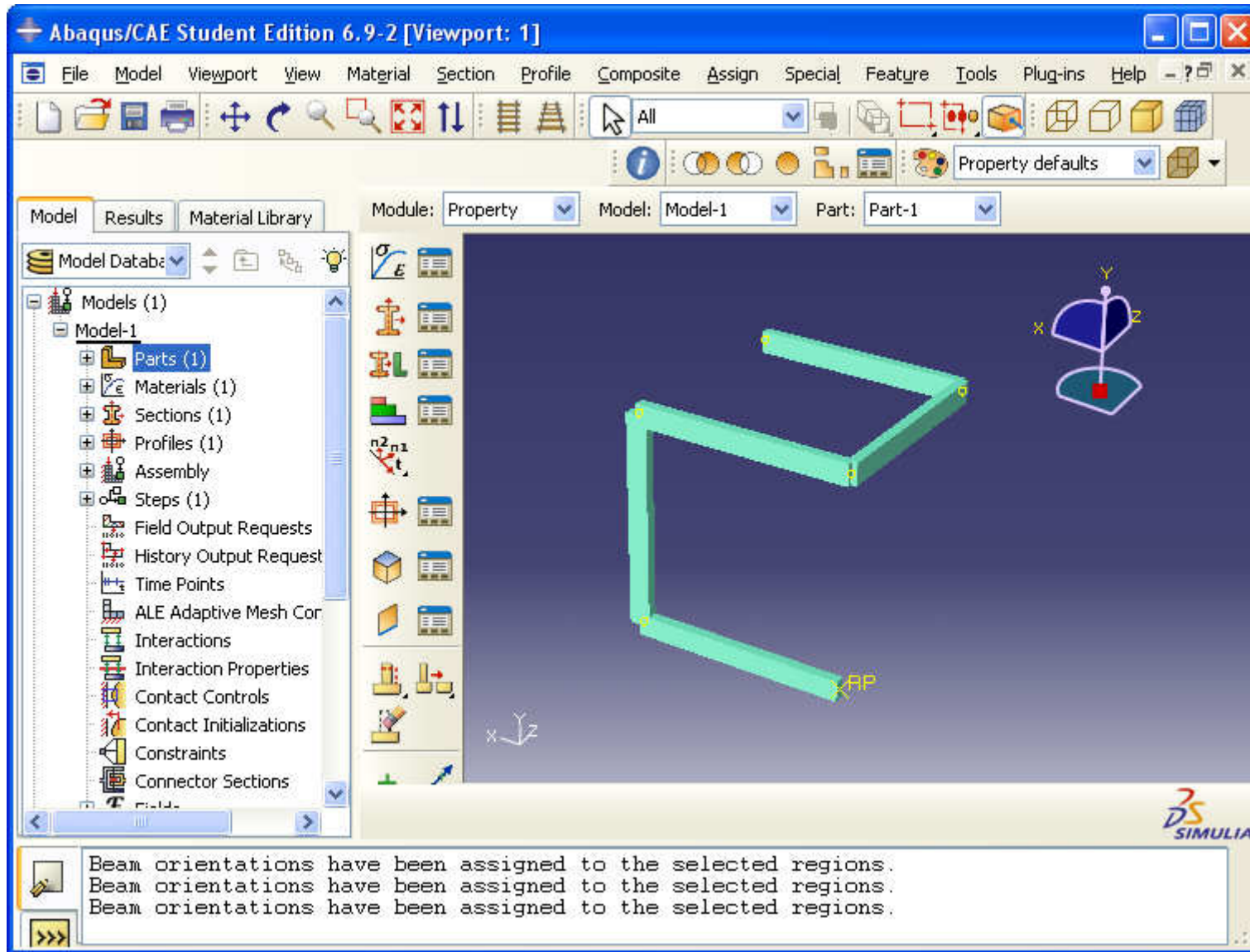


The main direction of the beam is not correctly set, in this case

# Assign beam orientation

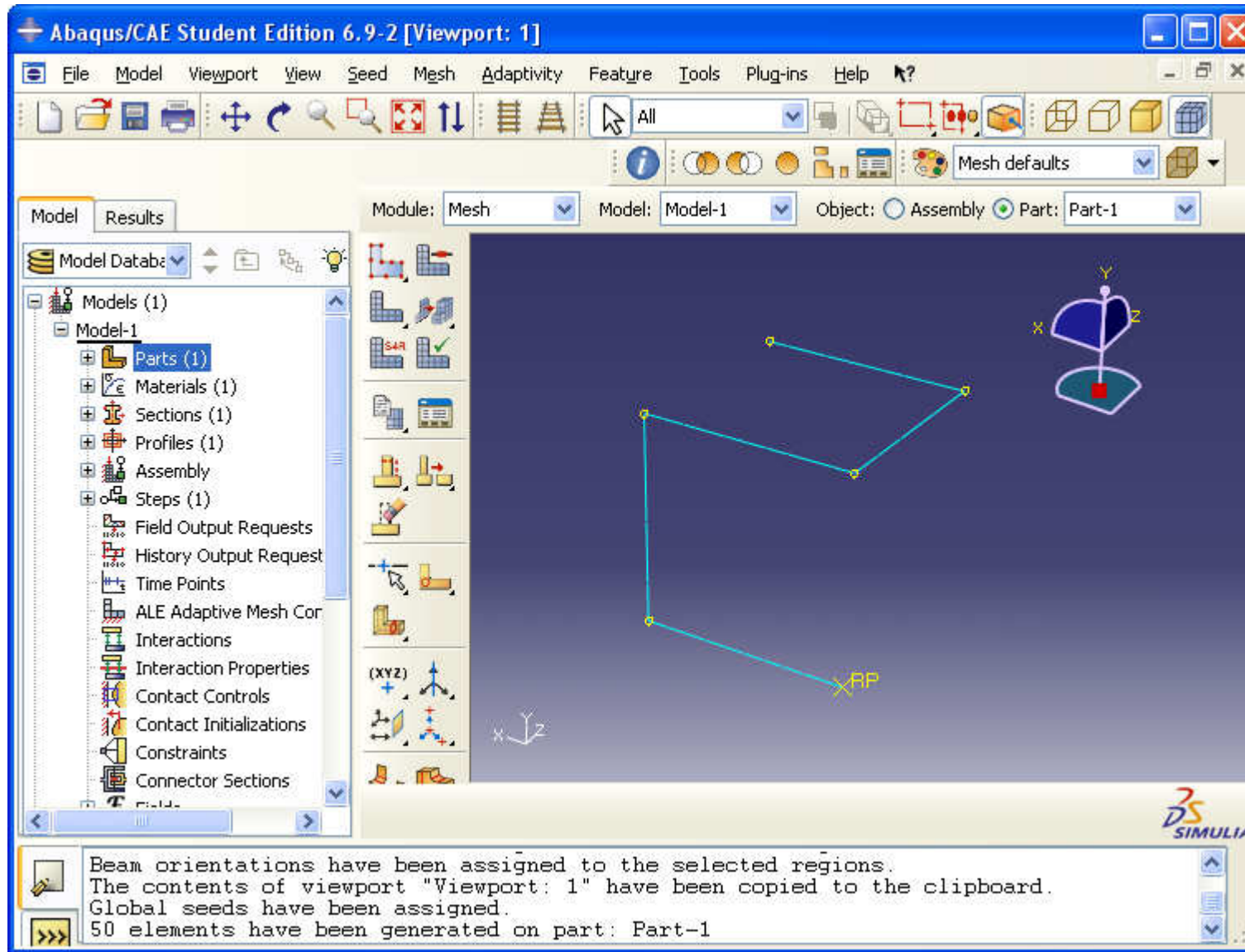


# The correct orientation of the beam





# Mesh creation



Turn off  
beam  
rendering go  
to mesh  
module and  
apply grid

# Boundary conditions - fixed

The screenshot displays the Abaqus/CAE Student Edition 6.9-2 interface. The main viewport shows a 3D model of a truss structure with a fixed boundary condition applied to one of its nodes, indicated by a red circle and a red arrow pointing to the 'Edit Boundary Condition' dialog box. The dialog box is titled 'Edit Boundary Condition' and contains the following information:

- Name: BC-1
- Type: Displacement/Rotation
- Step: Initial
- Region: (Picked)
- CSYS: (Global) [Edit...] [Create...]
- U1
- U2
- U3
- UR1
- UR2
- UR3

A note at the bottom of the dialog box states: "Note: The displacement value will be maintained in subsequent steps." The dialog box has 'OK' and 'Cancel' buttons.

The left-hand side of the interface shows the 'Model Database' tree with the following structure:

- Model Database
  - Sections (1)
  - Profiles (1)
  - Assembly
  - Steps (2)
    - Initial
      - Interactions
      - BCs (1)
        - BC-1 (Created)
      - Predefined Fields
    - Step-1
      - Field Output Request
      - History Output Request
      - ALE Adaptive Mesh Control
      - Interactions
      - Loads (1)
        - Load-1 (Created)
      - BCs (1)
      - Predefined Fields
      - Load Cases

The bottom status bar shows the following job history:

```
Job Job-1 aborted due to errors.  
The job input file "Job-1.inp" has been submitted for analysis.  
Job Job-1: Analysis Input File Processor completed successfully.  
Job Job-1: Abaqus/Standard completed successfully.  
Job Job-1 completed successfully.
```

# Boundary conditions - fixed

Abaqus/CAE Student Edition 6.9-2 [Viewport: 1]

File Model Viewport View Load BC Predefined Field Load Case Feature Tools Plug-ins Help

Model Results

Model Database

- Primitives (1)
- Assembly
- Steps (2)
  - Initial
    - Interactions
    - BCs (1)
    - Predefined Fields
  - Step-1
    - Field Output Request
    - History Output Request
    - ALE Adaptive Mesh Control
    - Interactions
    - Loads (1)
      - Load-1 (Created)
    - BCs (1)
    - Predefined Fields
    - Load Cases

- Field Output Requests (1)
- History Output Requests (1)

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

Assembly defaults

Fill out the Edit Load dialog

Beam orientations have been assigned to the selected regions.  
The contents of viewport "Viewport: 1" have been copied to the clipboard.  
Global seeds have been assigned.  
50 elements have been generated on part: Part-1

**Create Load**

Name: Load-1

Step: Step-1

Procedure: Static, General

Category:  Mechanical  Thermal  Acoustic  Fluid  Electrical

Types for Selected Step: **Concentrated force**, Moment, Pressure, Shell edge load, Surface traction, Pipe pressure

**Edit Load**

Name: Load-1

Type: Concentrated force

Step: Step-1 (Static, General)

Region: (Picked) [Edit Region...](#)

CSYS: (Global) [Edit...](#) [Create...](#)

Distribution: Uniform [Create...](#)

CF1: 0

CF2: -2

CF3: 0

Amplitude: (Ramp) [Create...](#)

Follow nodal rotation

**Note:** Force will be applied per node.

[OK](#) [Cancel](#)

# Results

