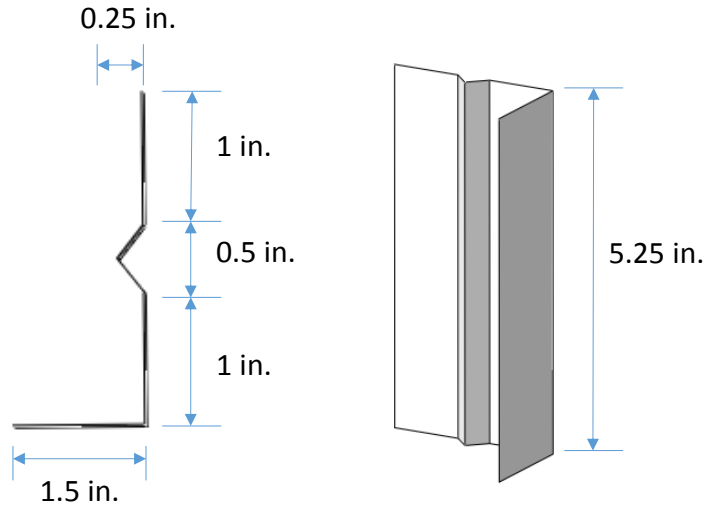


2015 International Student Competition on Cold-Formed Steel Design

Design Example



Nominal shear strength:

From ABAQUS, we obtained the elastic buckling load per unit length: 1.1127 kip/in. By multiplying the ABAQUS result with the angle's depth, we can get the elastic buckling shear force.

$$V_{cr} = \left(1.1127 \frac{\text{kip}}{\text{in.}}\right) \times (5.25 \text{ in.}) = 5.842 \text{ kips}$$

$$F_{cr} = \frac{V_{cr}}{Bt} = \frac{5.842}{5.25 \times 0.0451} = 24.67 \text{ ksi}$$

$$\lambda = \sqrt{\frac{F_y}{F_{cr}}} = \sqrt{\frac{33}{24.67}} = 1.157$$

$$V_n = \text{minimum} (0.17\lambda^{-0.8}F_y Bt, \quad 0.35F_y Bt)$$

$$= \text{minimum} \{0.17(1.157)^{-0.8}(33\text{ksi})(5.25\text{in.})(0.0451\text{in.}), \quad 0.35(33\text{ksi})(5.25\text{in.})(0.0451\text{in.})\}$$

$$= \text{minimum} (1.182 \text{ kips}, 2.735 \text{ kips})$$

$$= 1.182 \text{ kips} = 1182 \text{ lbf}$$

The total weight:

$$W = (4.207 \text{ in.})(5.25 \text{ in.})(0.0451 \text{ in.}) \left(0.284 \frac{\text{lb}}{\text{in.}^3}\right) = 0.2829 \text{ lb}$$

Nominal shear strength per unit weight:

$$\frac{V_n}{W} = \frac{1182}{0.2829} = 4178 \text{ lbf/lb}$$

The following pages include the screen shots of the step-by-step procedure of the ABAQUS modeling and analysis. The ABAQUS student edition can be obtained from 3DS Academy website

<http://academy.3ds.com/software/simulia/abaqus-student-edition/>

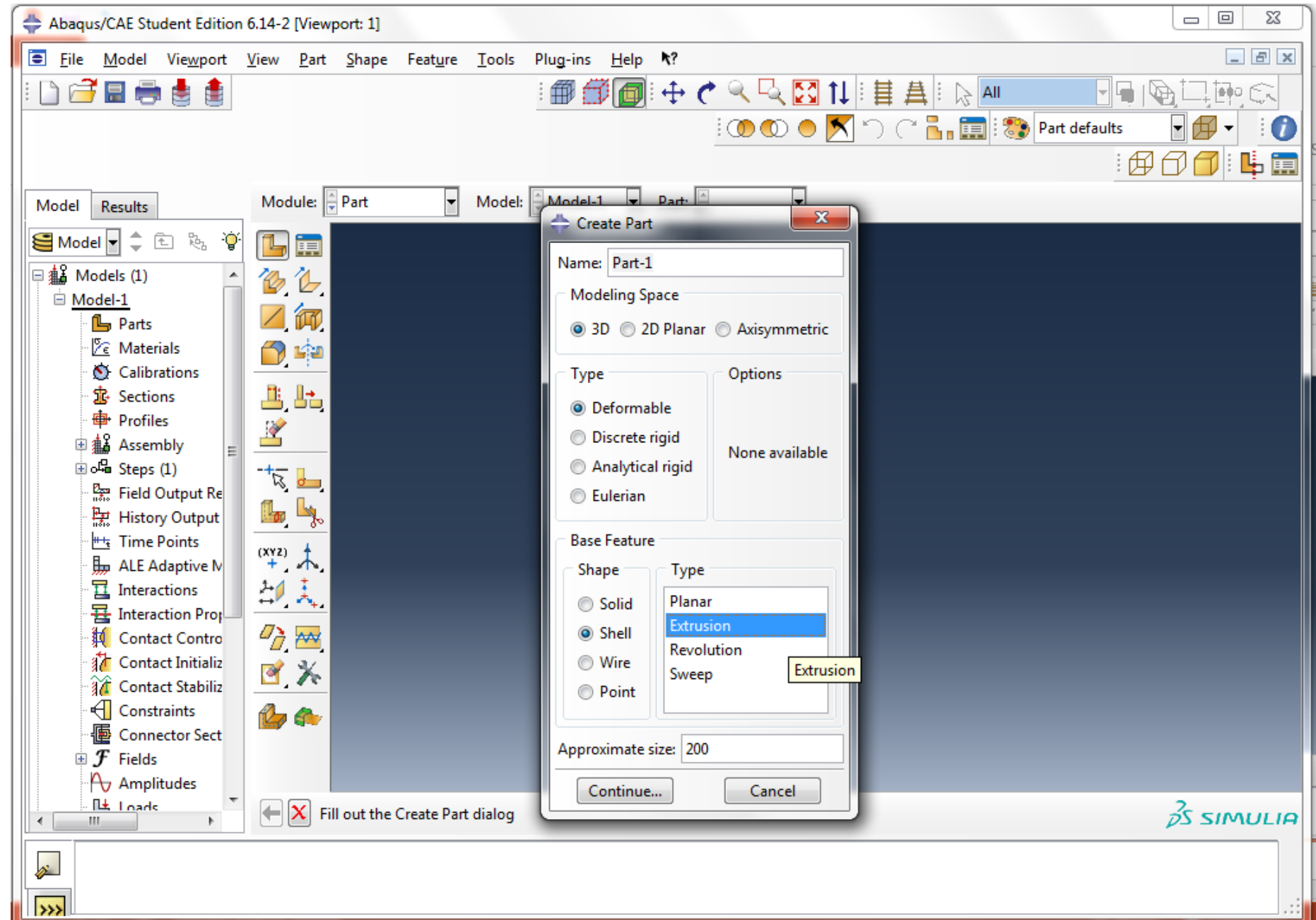
Step 1: create part



We will use shell elements for the clip angle model.

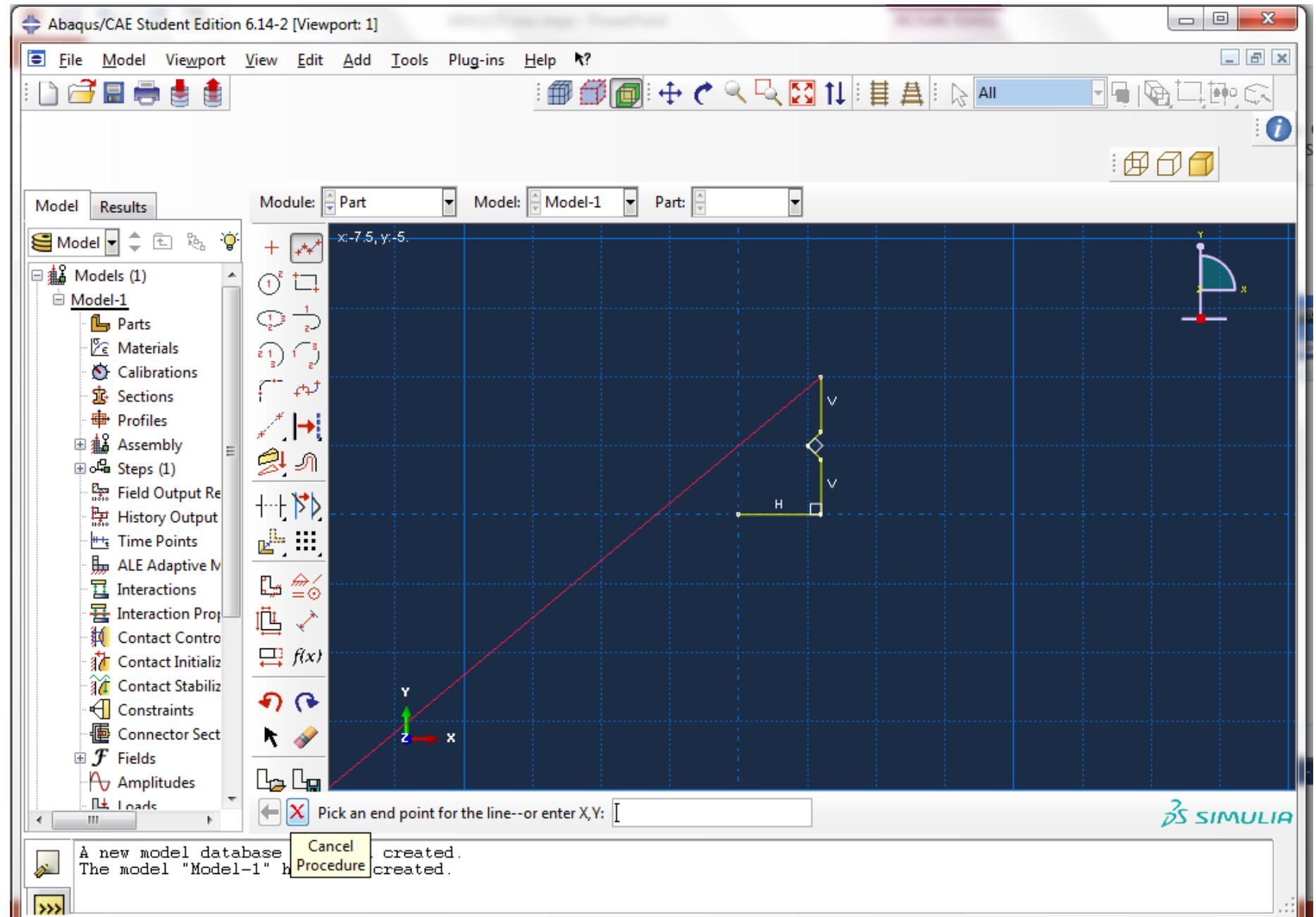
We create the 3D model using extrusion from a shell shape.

Start with the “Part”
Modulus

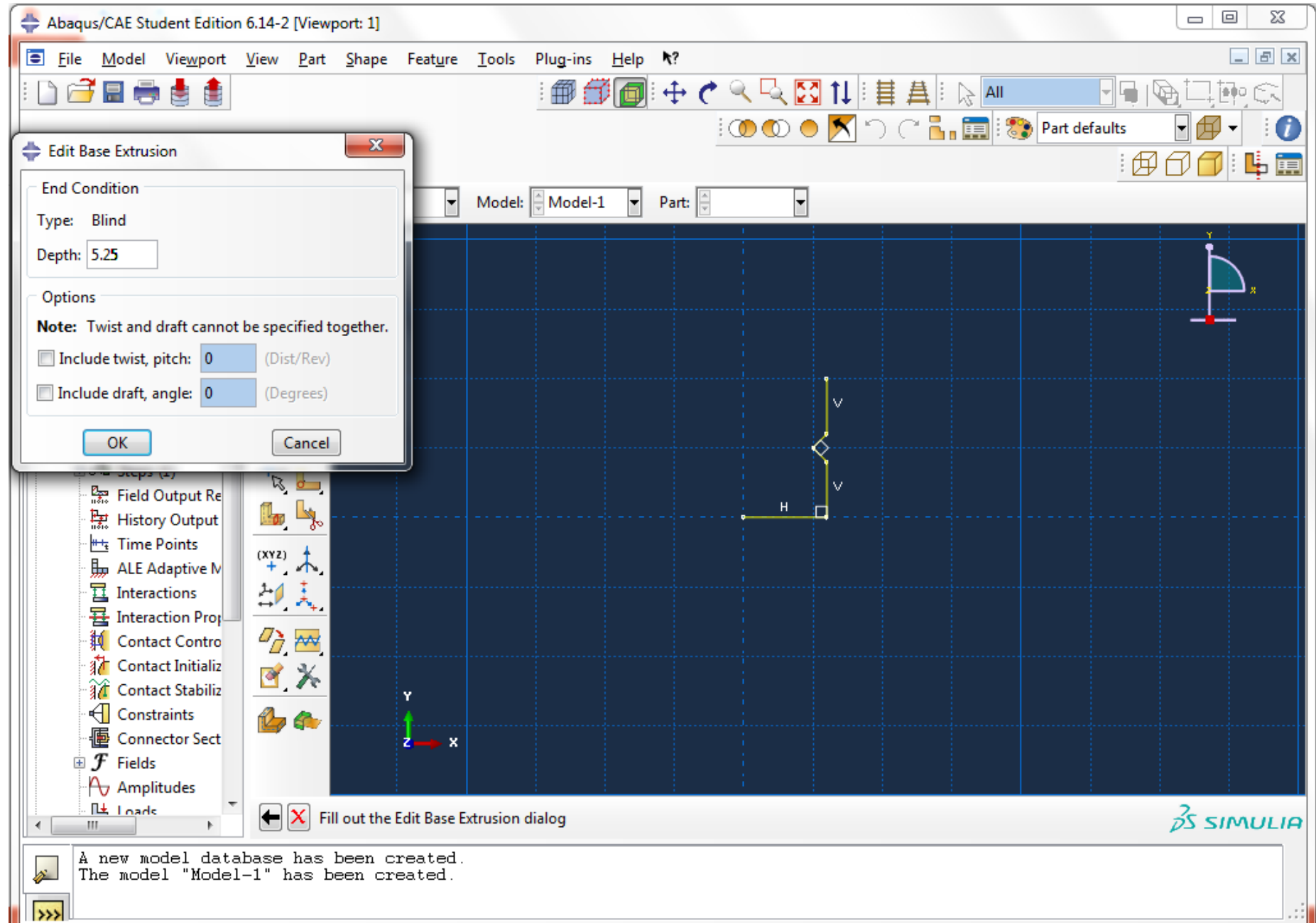
Click  to create part



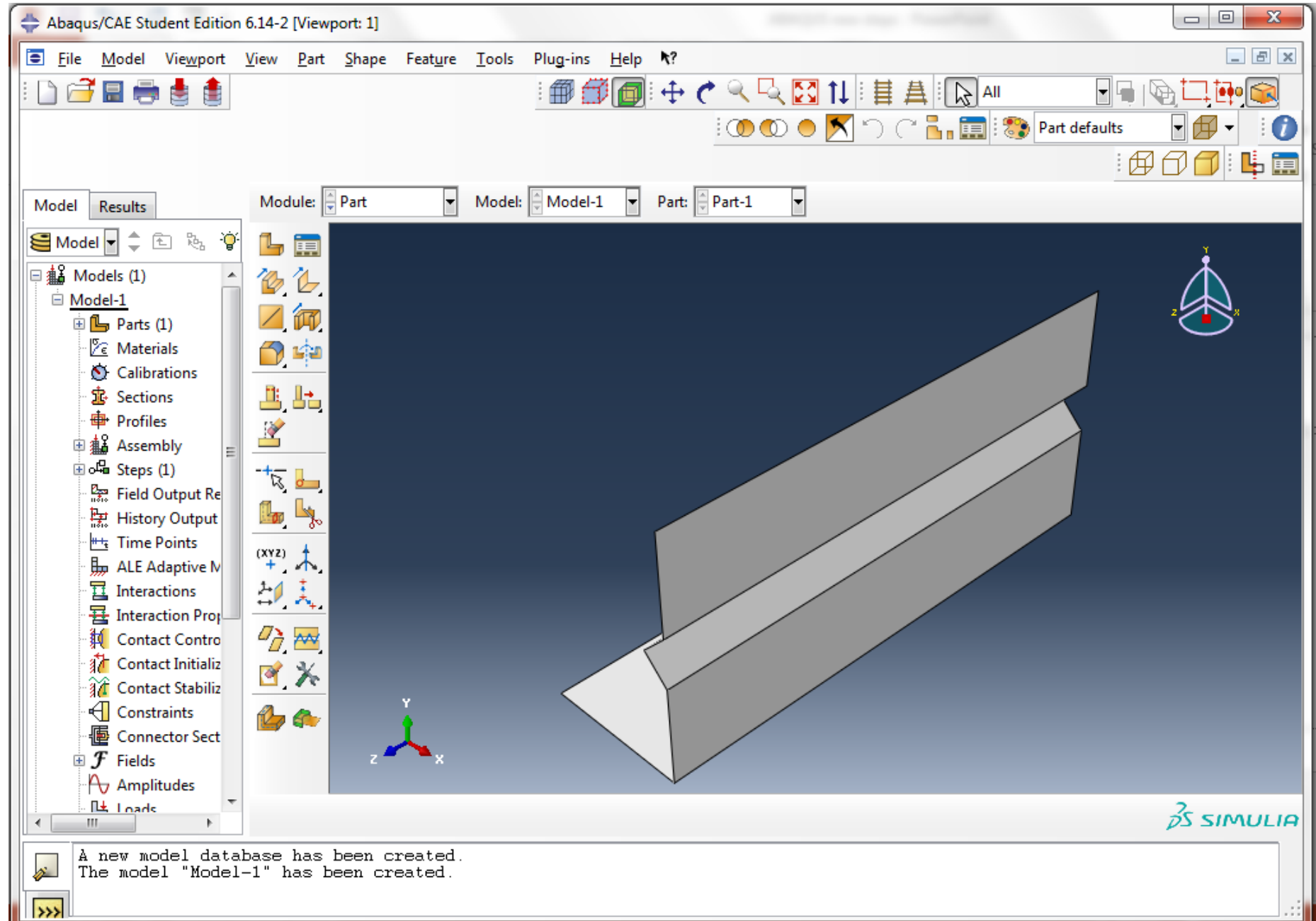
Click to  create the cross section shape.
Click  to finish.



Input the clip angle's length
5.25 (in.)



Step 1 finished.

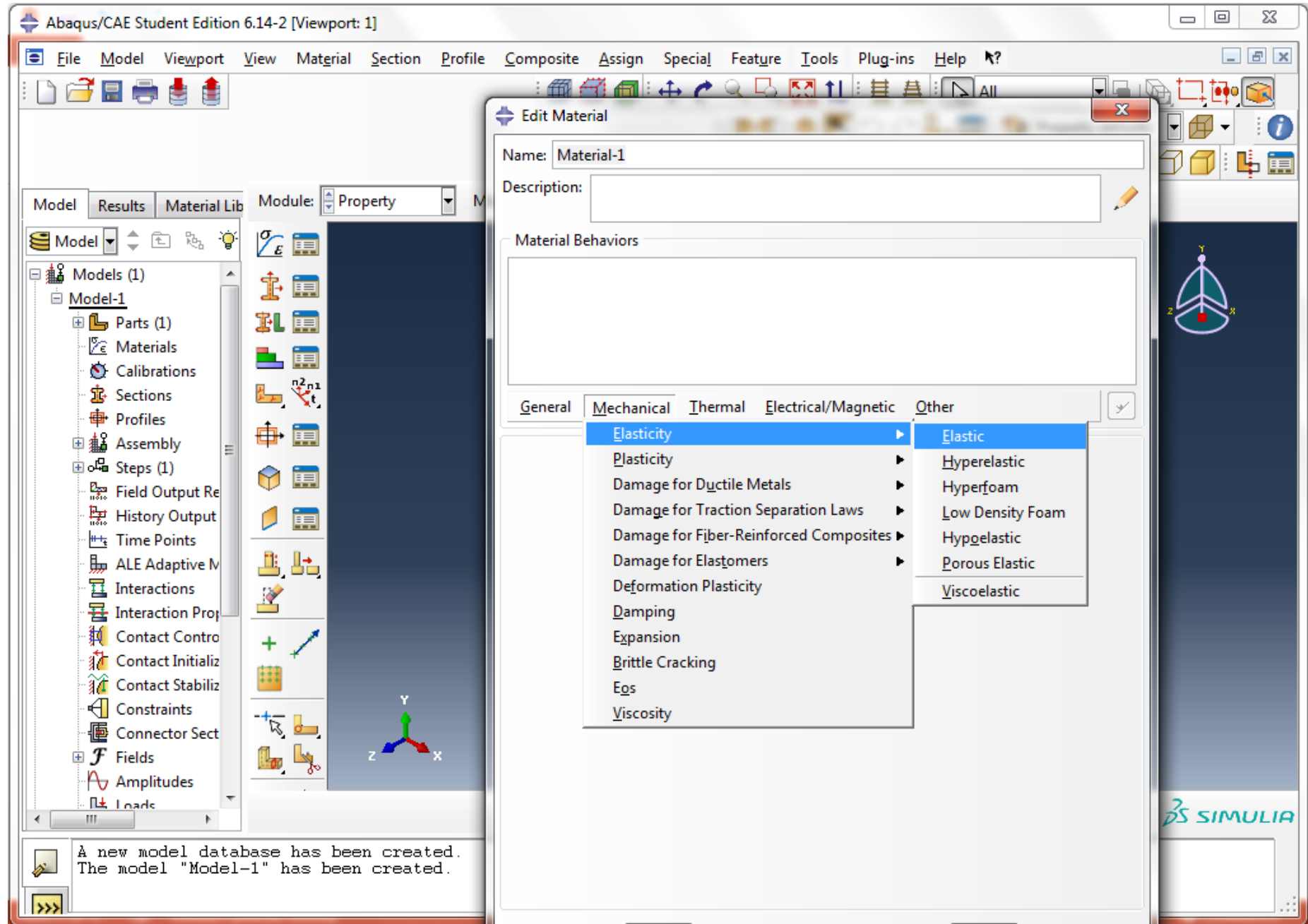


Step 2 create material model and section

Choose the “Property” Modulus

Choose “Mechanical”->”Elasticity”->Elastic to model the material properties.

We are doing an elastic buckling analysis here.



Input Young's modulus,
29500 (ksi);
Poisson's ratio 0.3

The screenshot displays the Abaqus/CAE Student Edition 6.14-2 interface. The 'Edit Material' dialog box is open, showing the following settings:

- Name: Material-1
- Description: (empty)
- Material Behaviors: Elastic
- General tab selected
- Elastic Type: Isotropic
- Use temperature-dependent data:
- Number of field variables: 0
- Moduli time scale (for viscoelasticity): Long-term
- No compression:
- No tension:


The 'Data' table in the dialog is as follows:

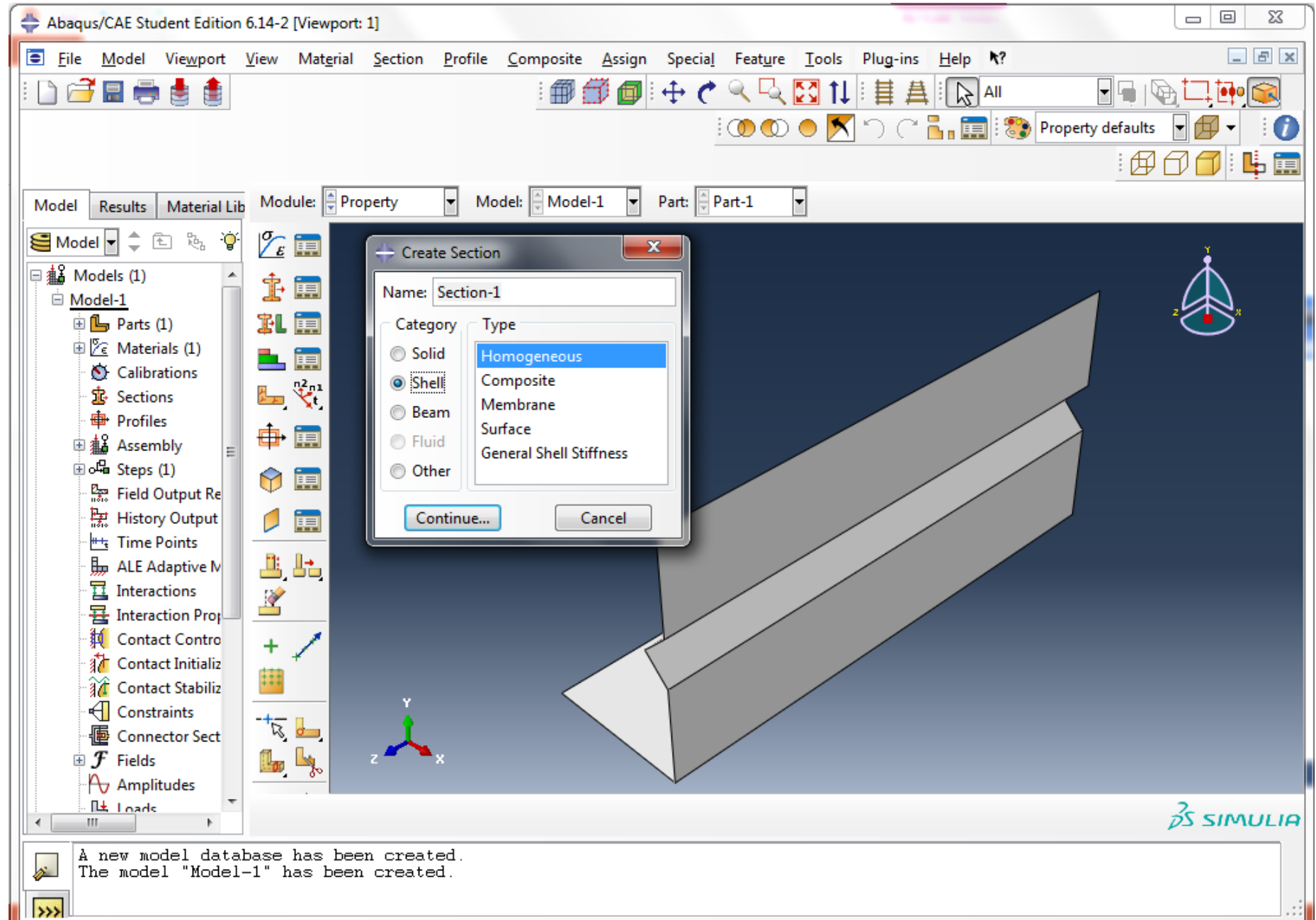
	Young's Modulus	Poisson's Ratio
1	29500	0.3

The background interface shows the 'Material Lib' tab selected in the top bar, and the 'Property' module selected in the 'Module' dropdown. The left tree view shows 'Model-1' expanded, with 'Materials' selected. A 3D coordinate system (X, Y, Z) is visible at the bottom center of the viewport.

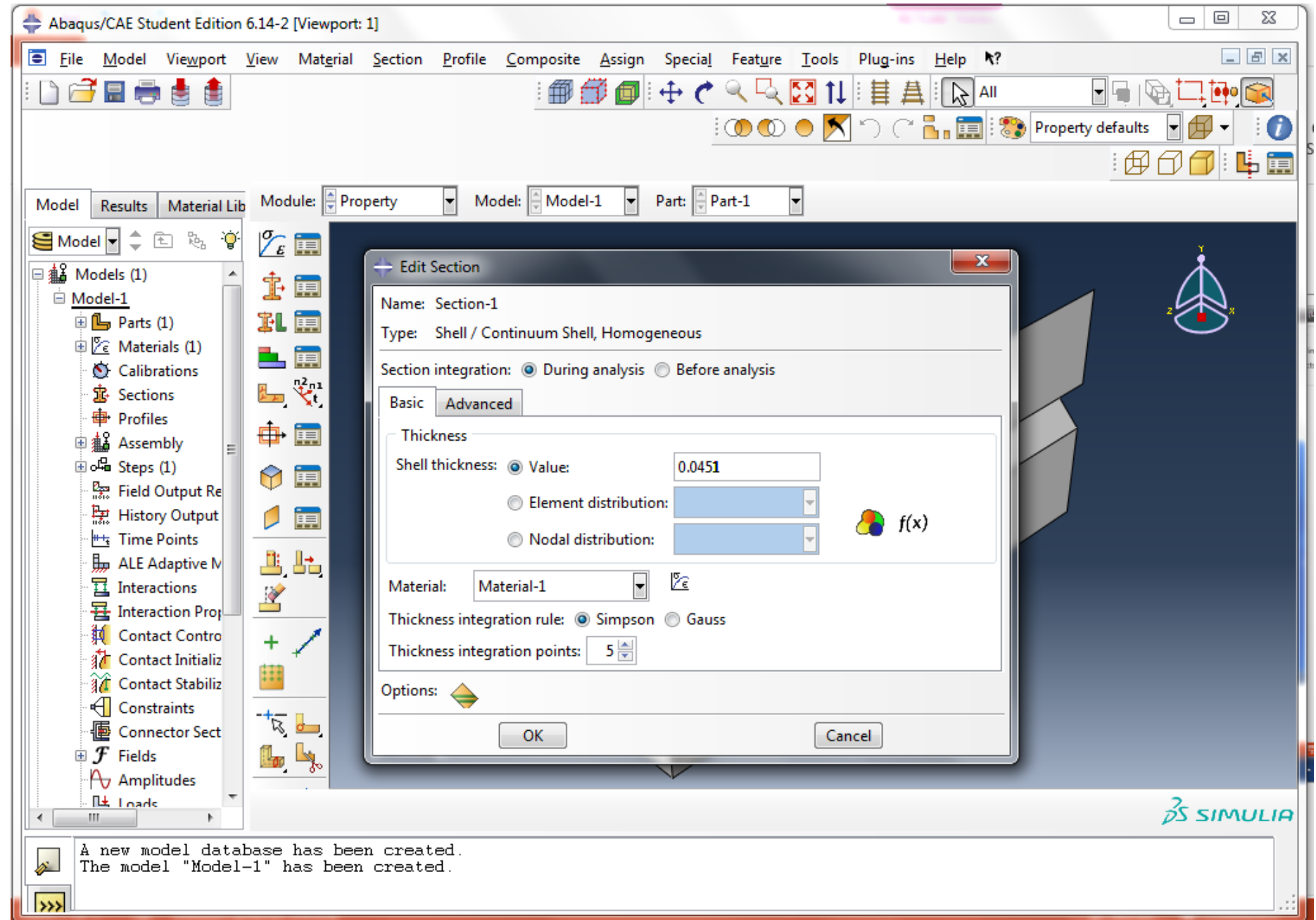
A message box at the bottom left of the interface reads: "A new model database has been created. The model 'Model-1' has been created."




Click  to create a section using “Shell” category and “Homogeneous” type.



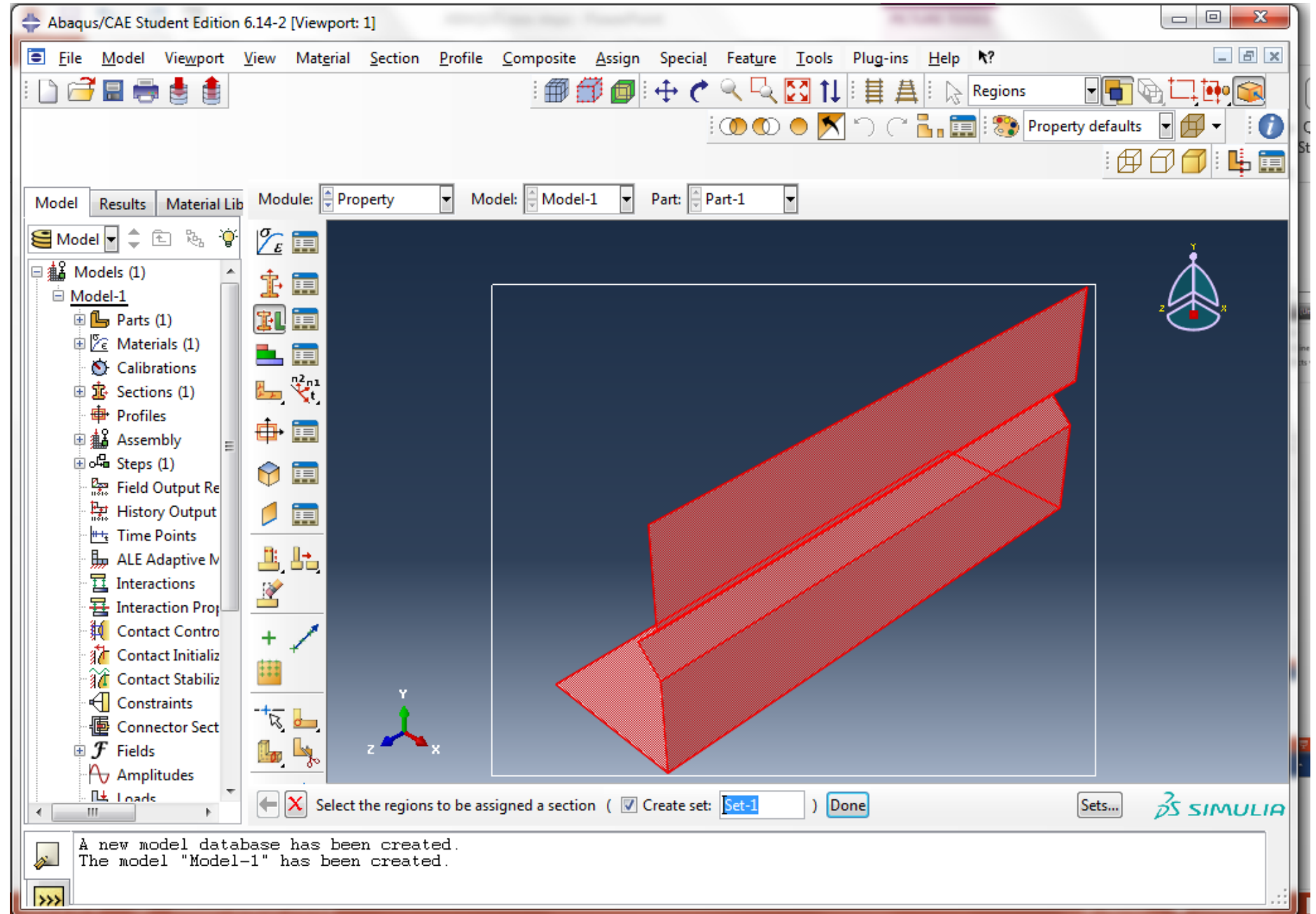
We define the thickness, 0.0451 (in.) in the section property



Click  to assign the newly created section to the clip angle part.


Select the entire clip angle.

Step 2 finished.

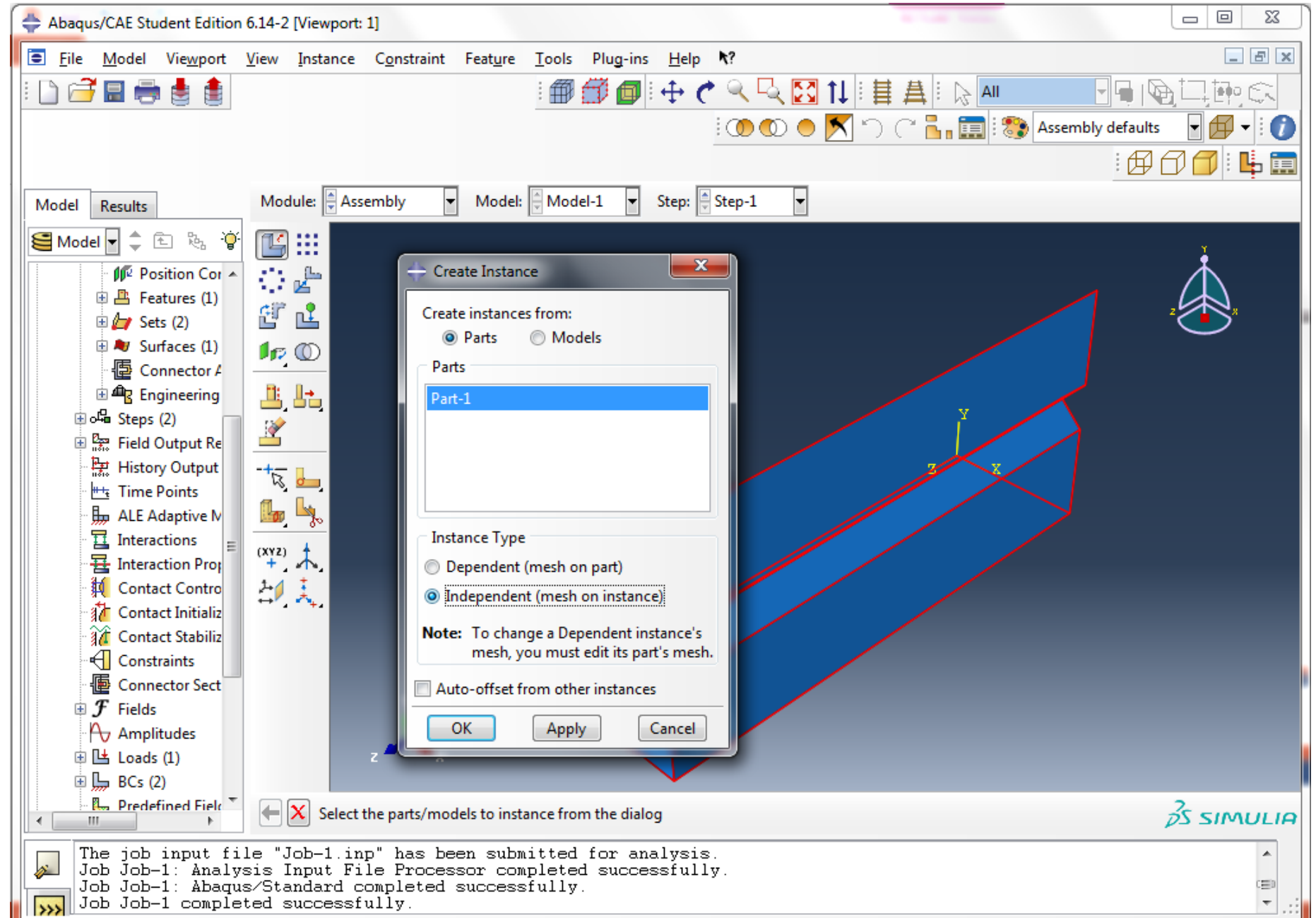


Step 3 create instance

In the “Assembly” Module


Click  to create an instance using the “Independent (mesh on instance)” Type.

Step 3 finished.

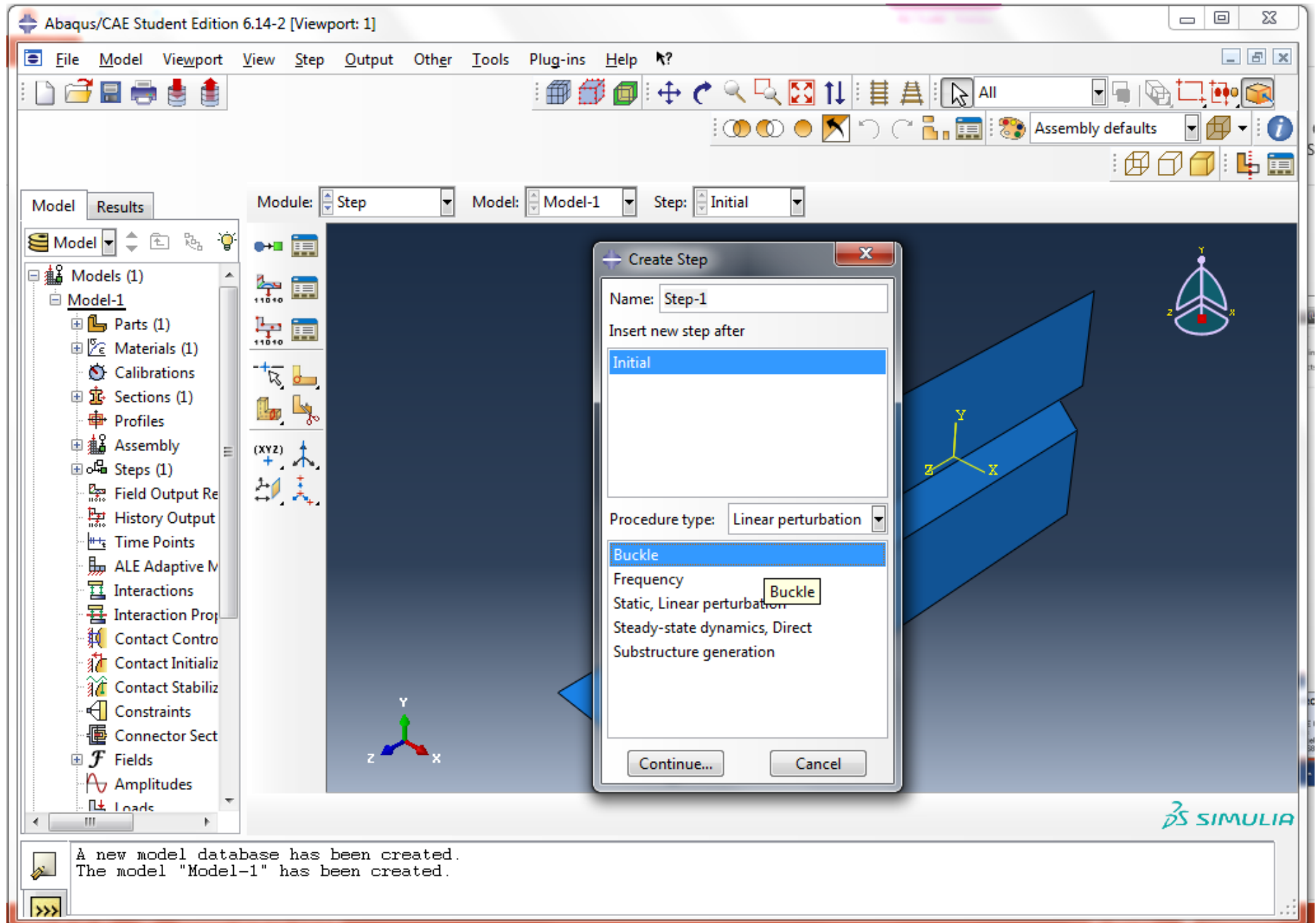


Step 4 create steps

In the “Step” Module

Click  to create step

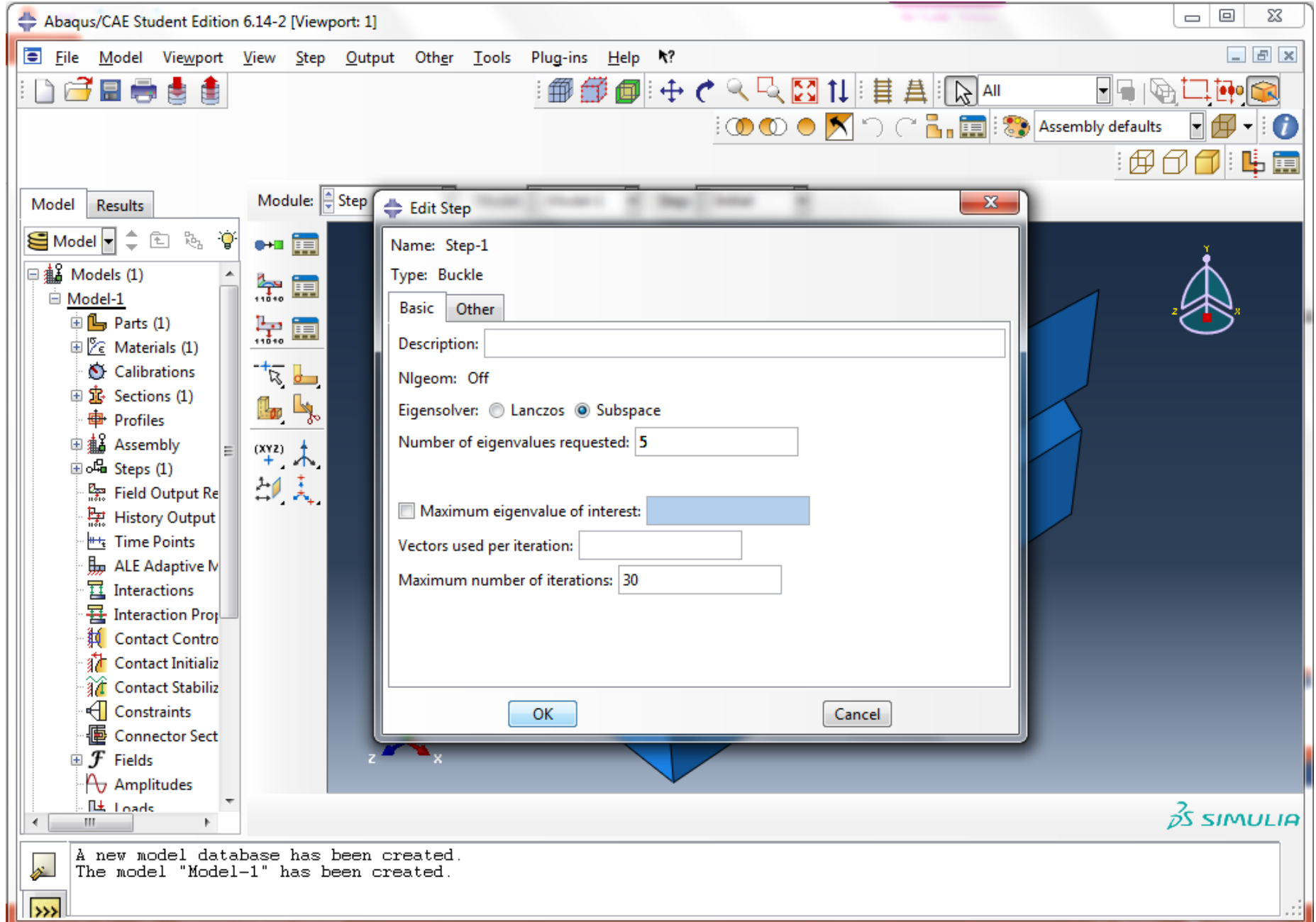
Since we are doing an elastic buckling/eigenvalue analysis, we use “Linear perturbation” as Procedure type and “Buckle” as the analysis type.



Continue on the step creation, we use "Subspace" and request "5" eigenvalues.


You can use other appropriate approaches, we focus on the 1st buckling mode in this analysis.

Step 4 finished




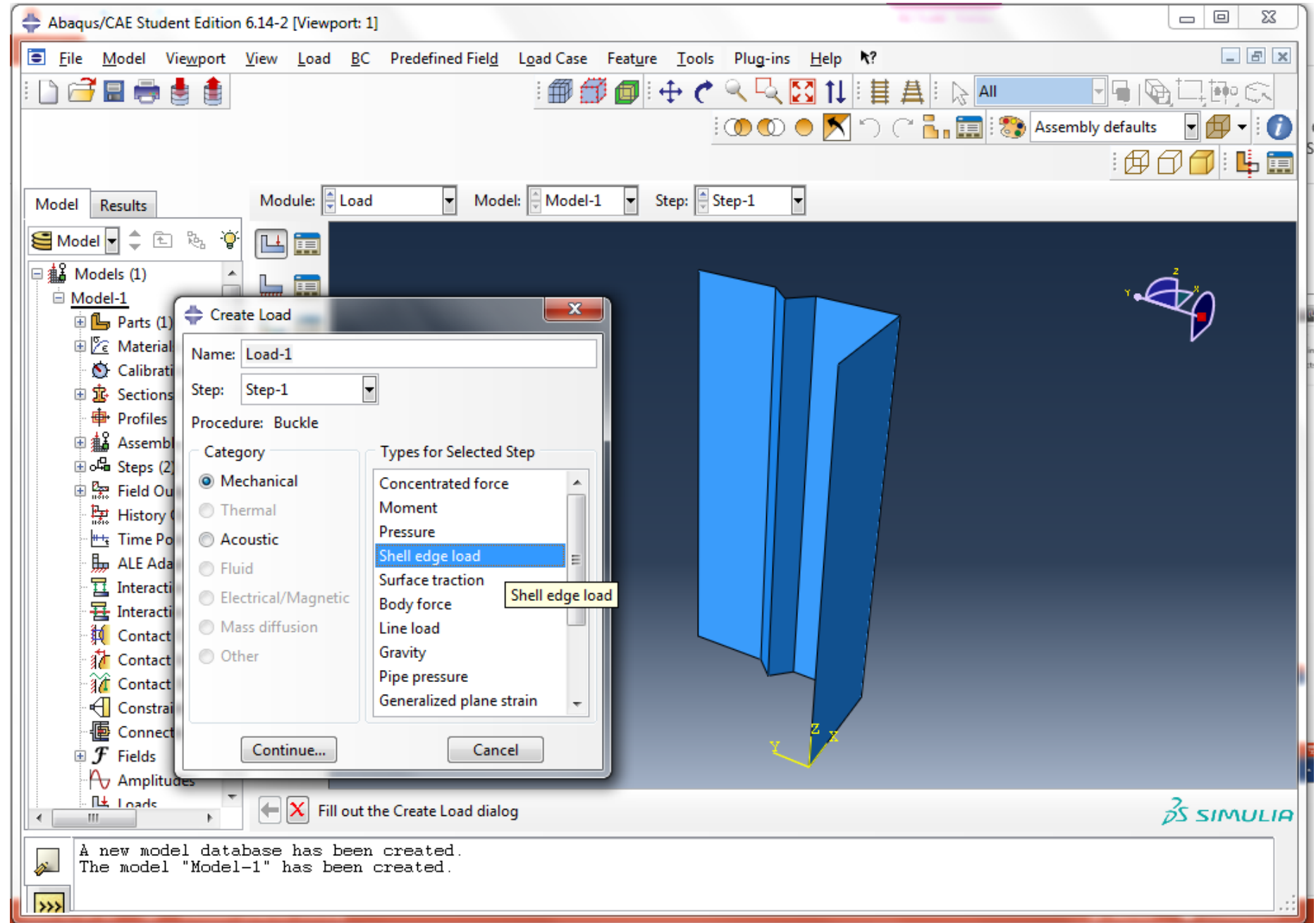
Step 5 create load and boundary conditions

In the “Load” module

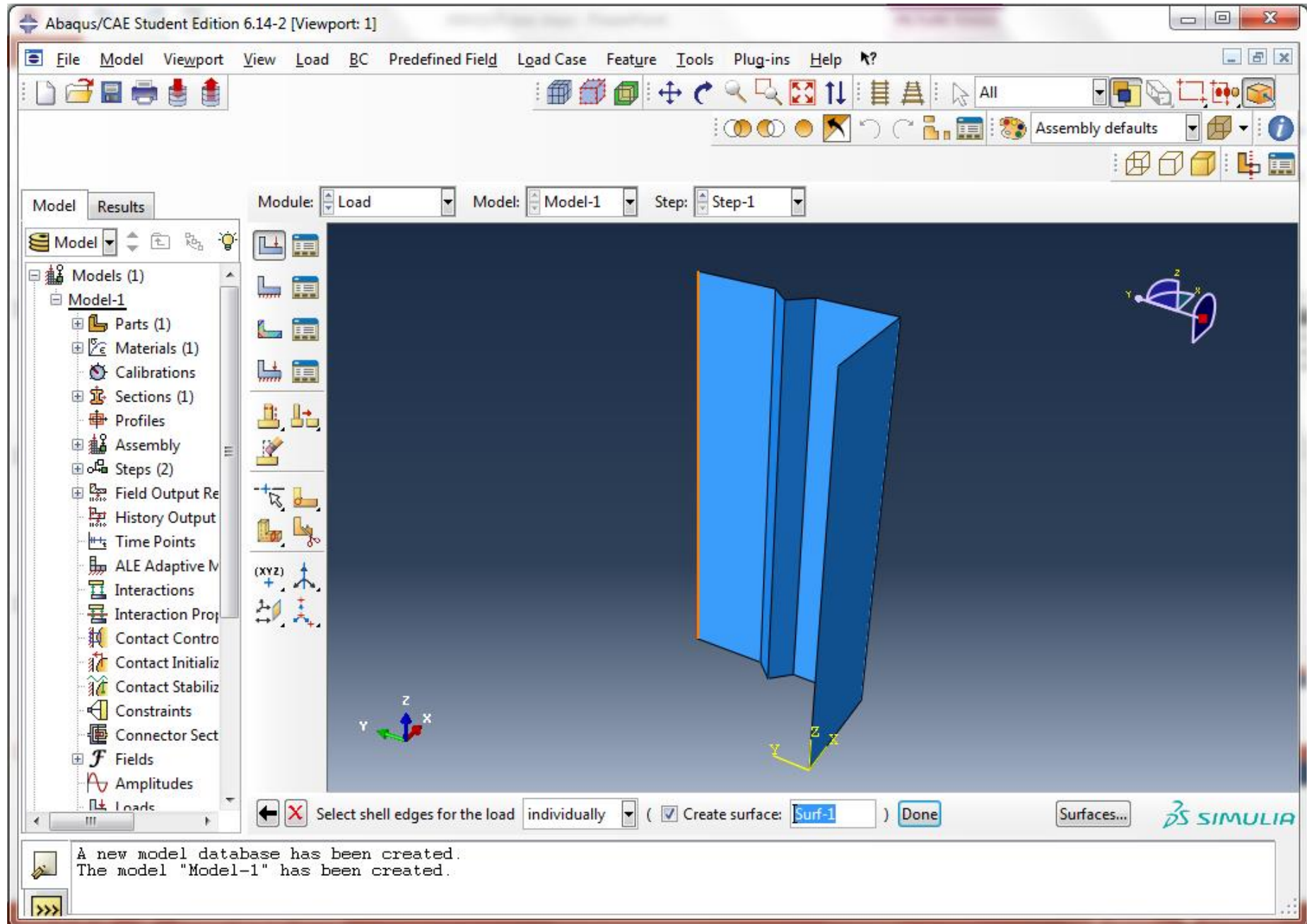
Click  to create the uniform shear force on the edge of cantilevered leg.

We use “Shell edge load.”

Use can click  to rotate the clip angle to a more practical view.

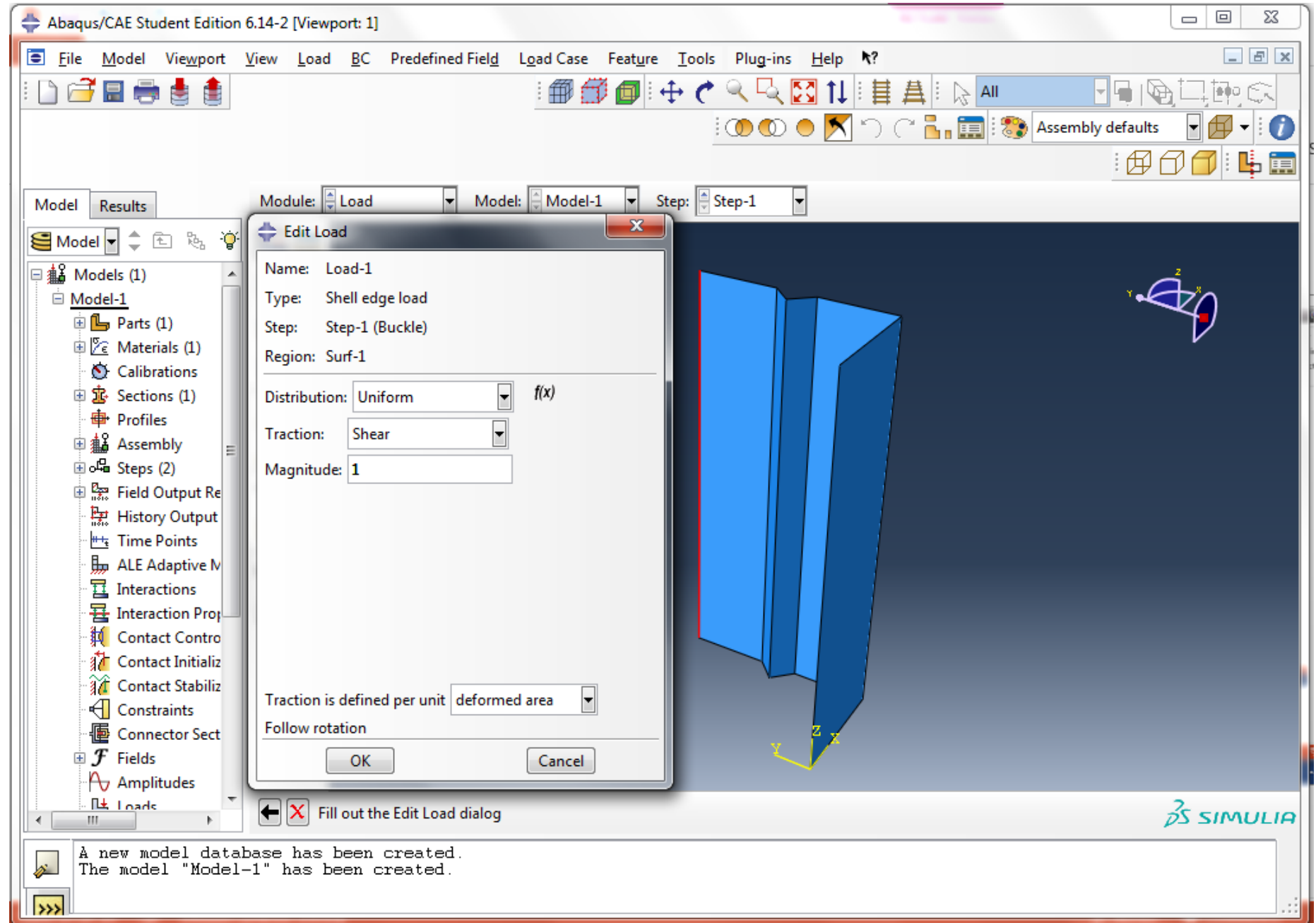



Select the edge of the cantilevered leg for the applied shear load

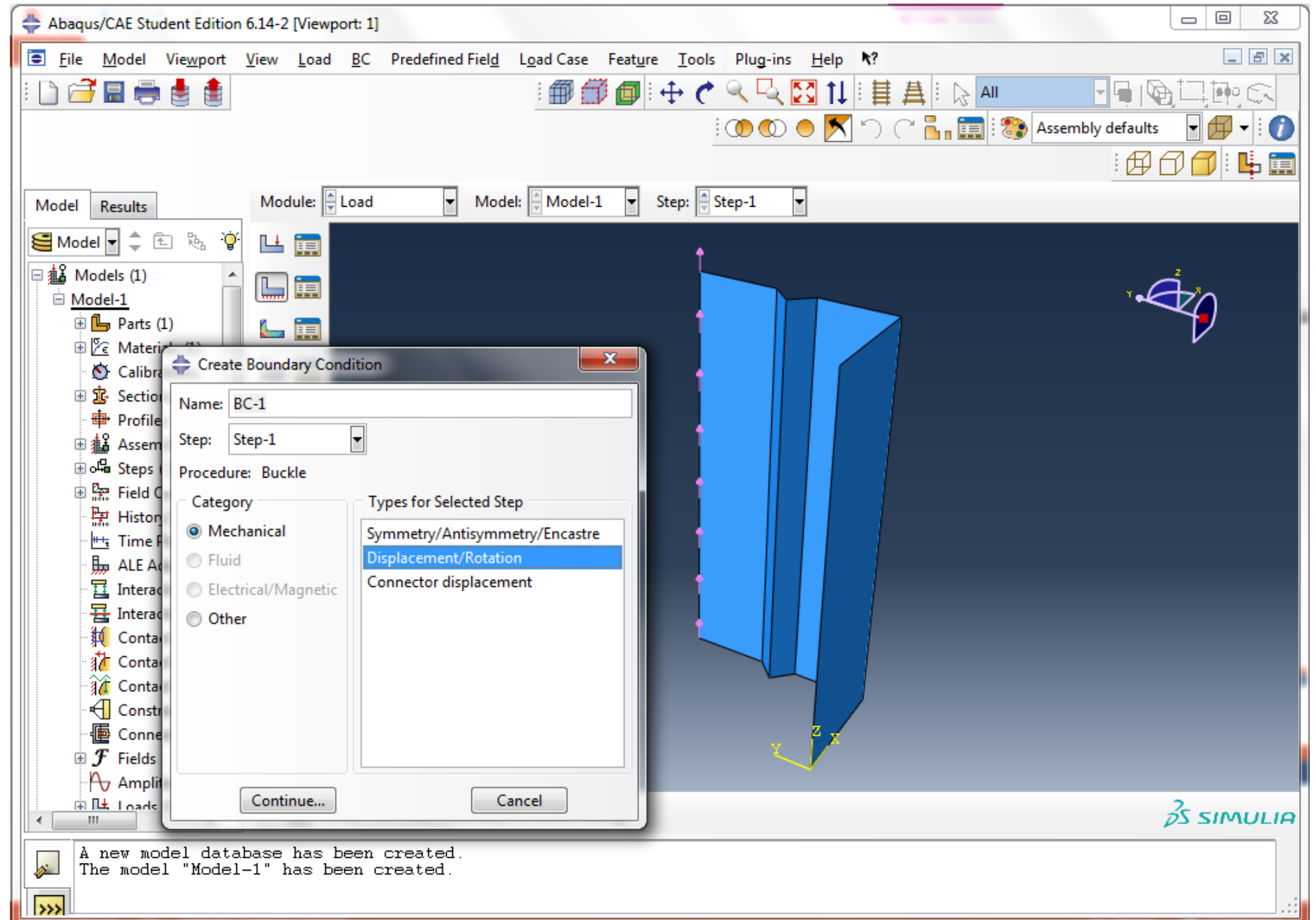


Specify the shell edge load:

“Uniform” distribution
“Shear” Traction
“1 (kips/in.)” Magnitude

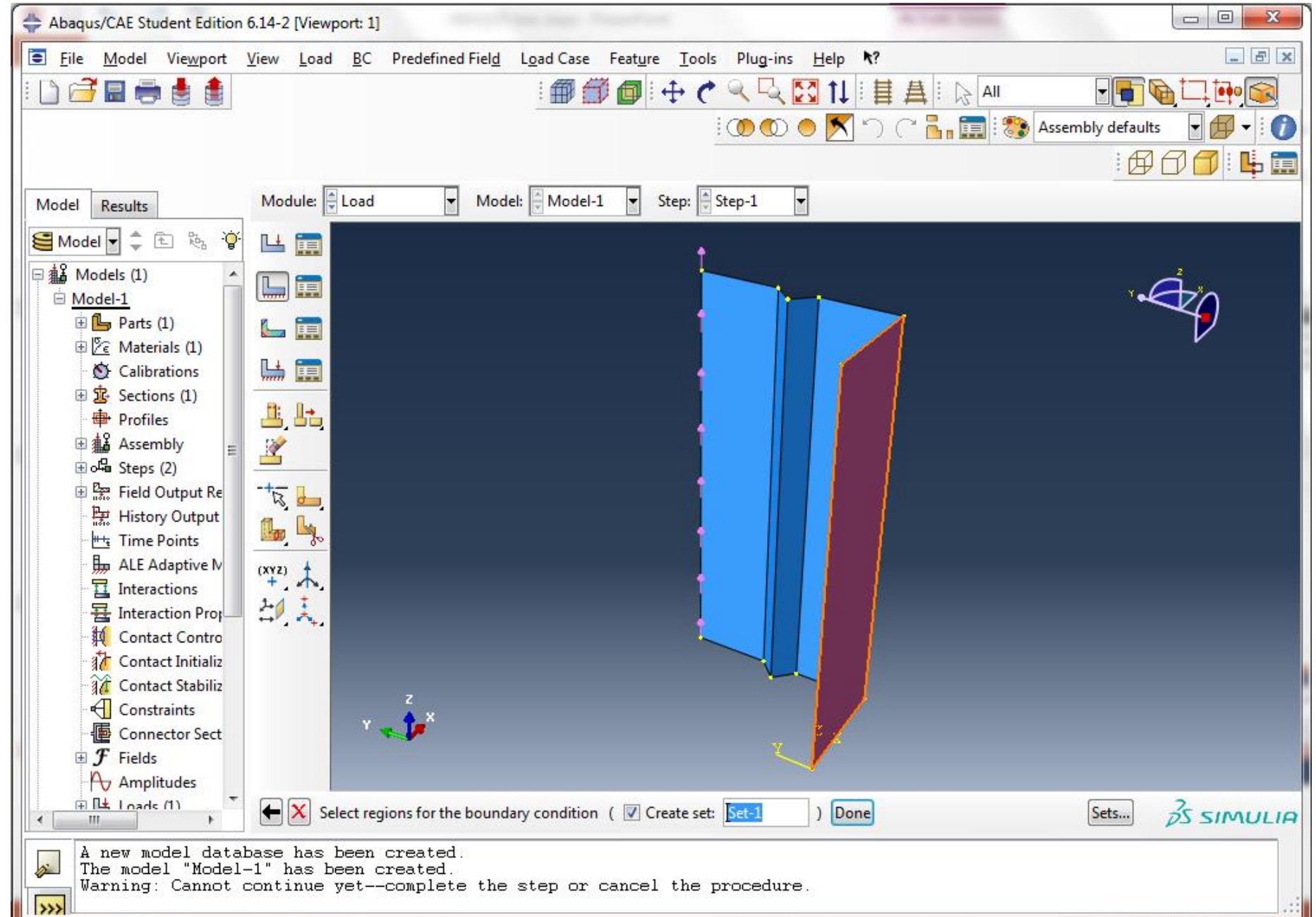


Click  to define boundary conditions. We use “Displacement/Rotation” type.



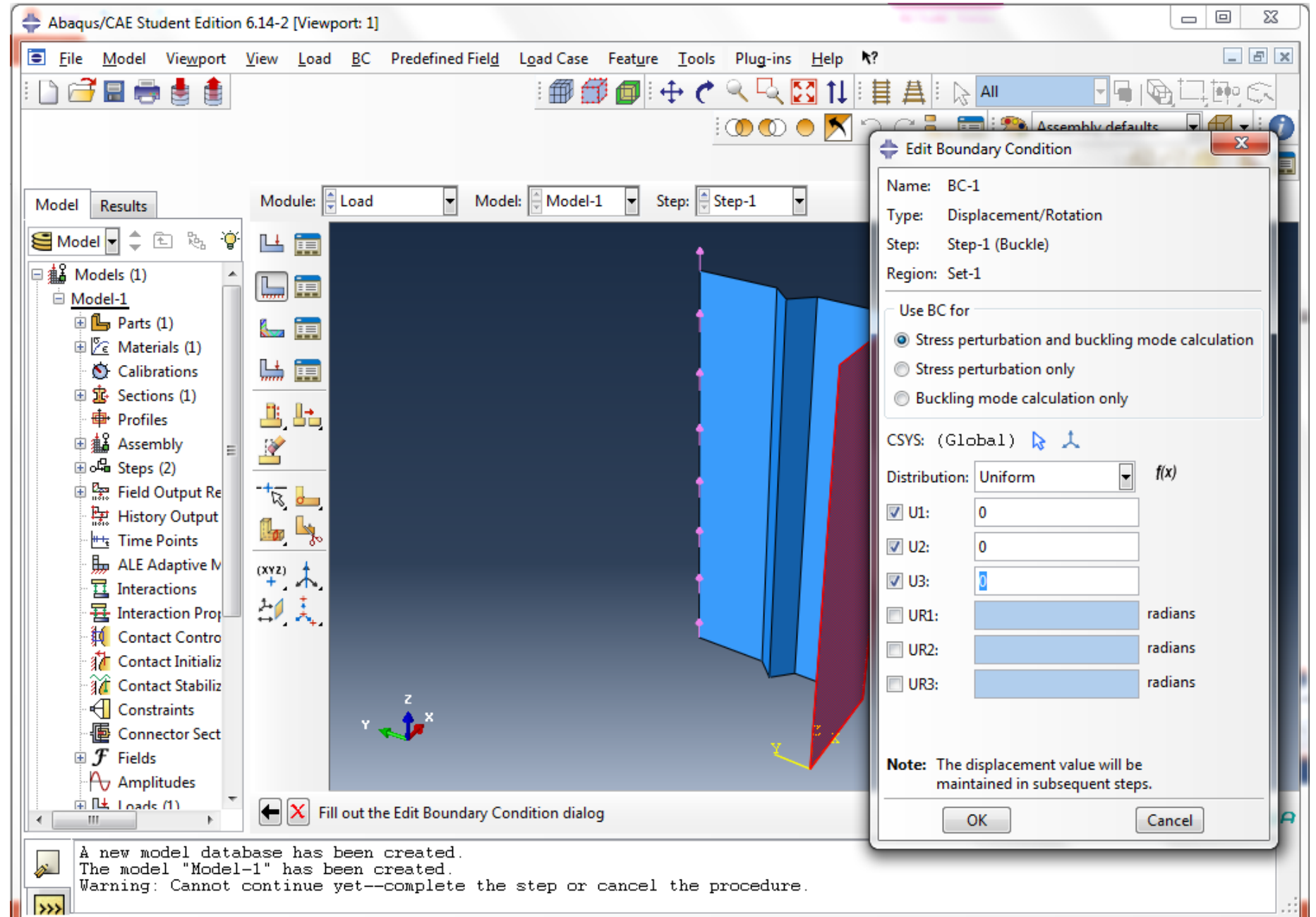
The first boundary condition is the fixed surface of the anchored leg.

Select the bottom surface of the anchored leg.



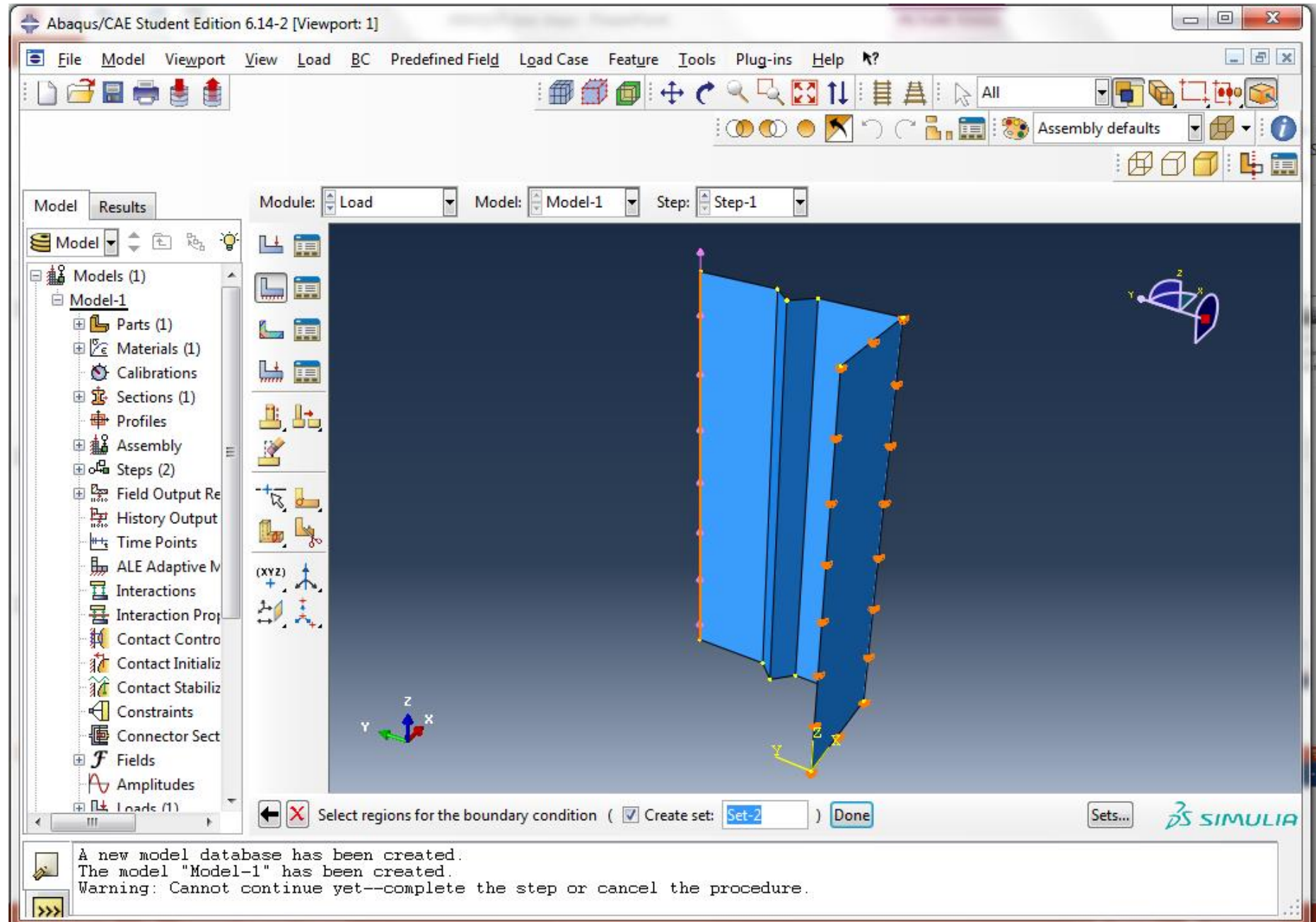
We fix the displacement of the surface in all 3 directions.

$U_1=0$
 $U_2=0$
 $U_3=0$



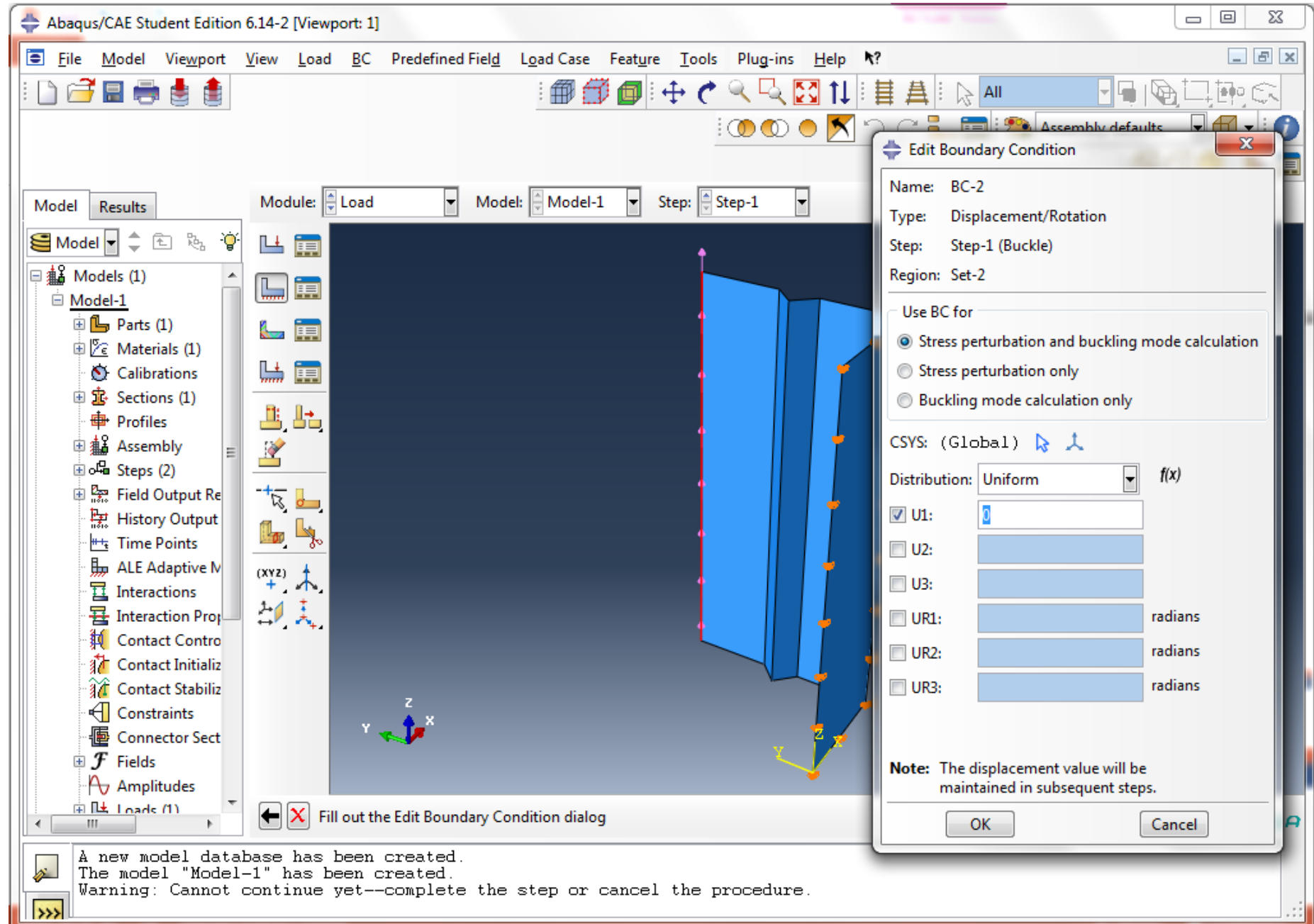
The 2nd boundary condition (BC) is the lateral constraint on the loaded edge.

Same step as BC1, select the edge of the cantilevered leg for BC2.




For the BC2, we only fix the out-of-plane movement
 $U_1=0$

Step 5 completed.

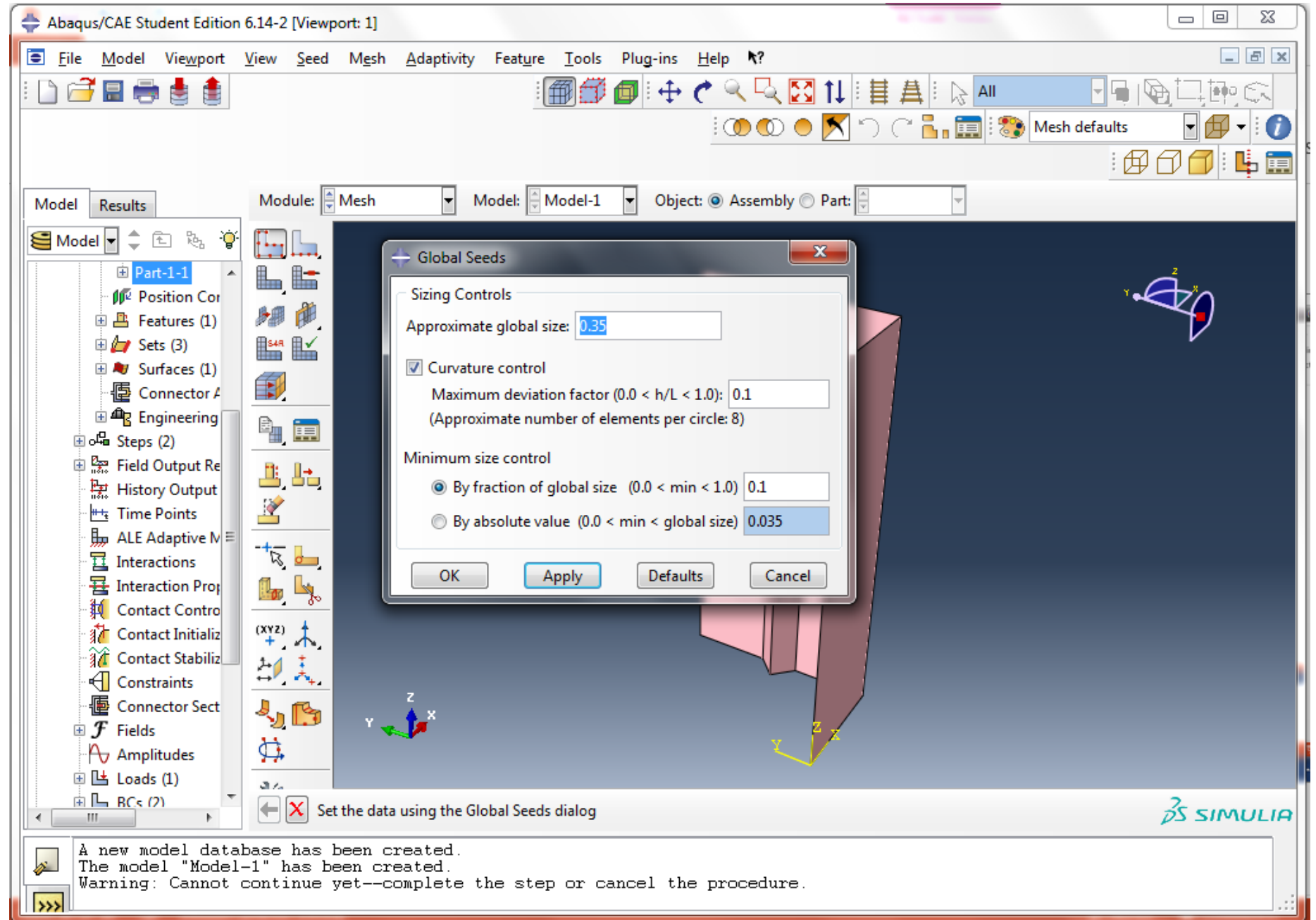


Step 6 create mesh

In the “Mesh” Module

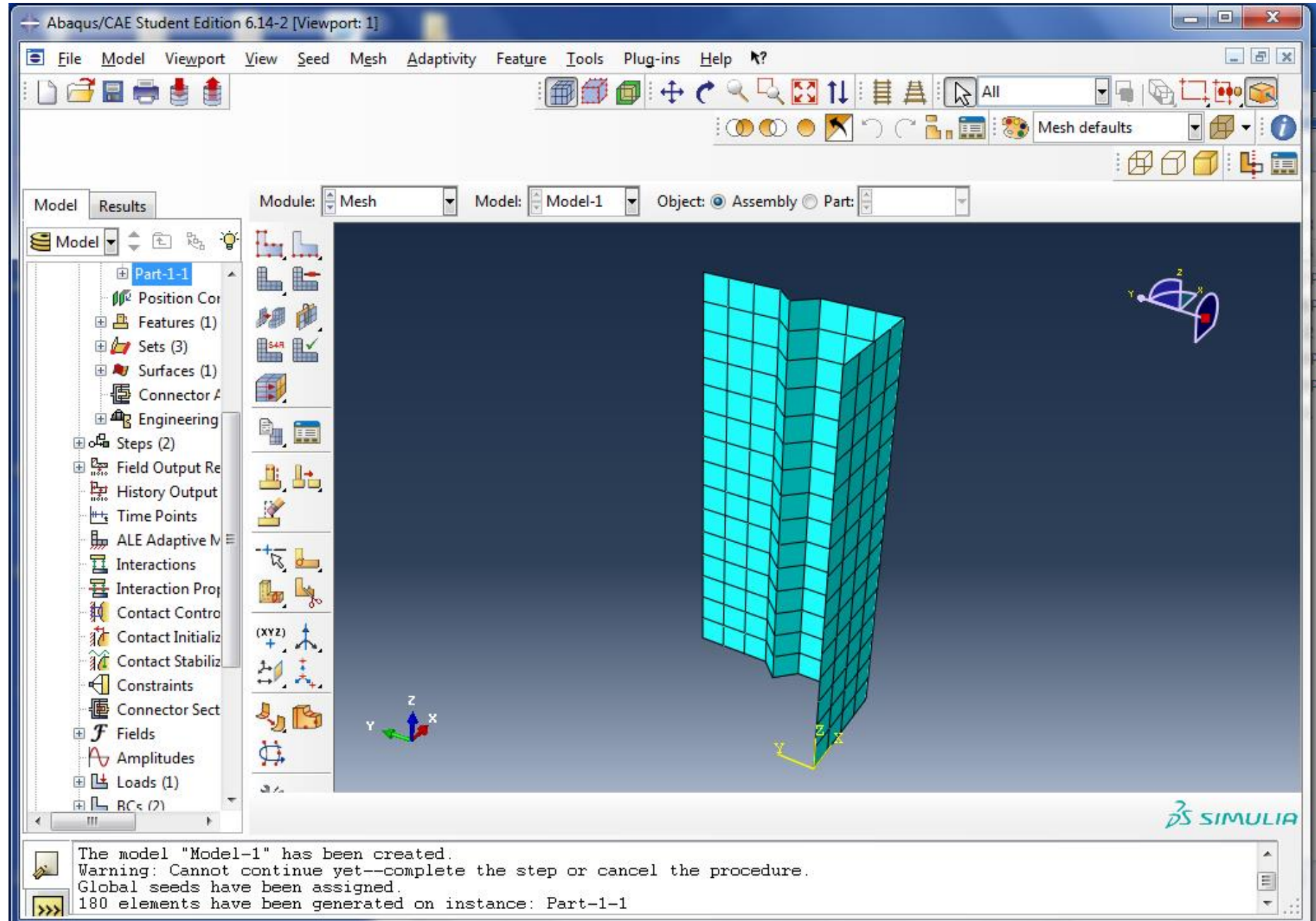
Click  to define the mesh sizes.

You can use the default setting.

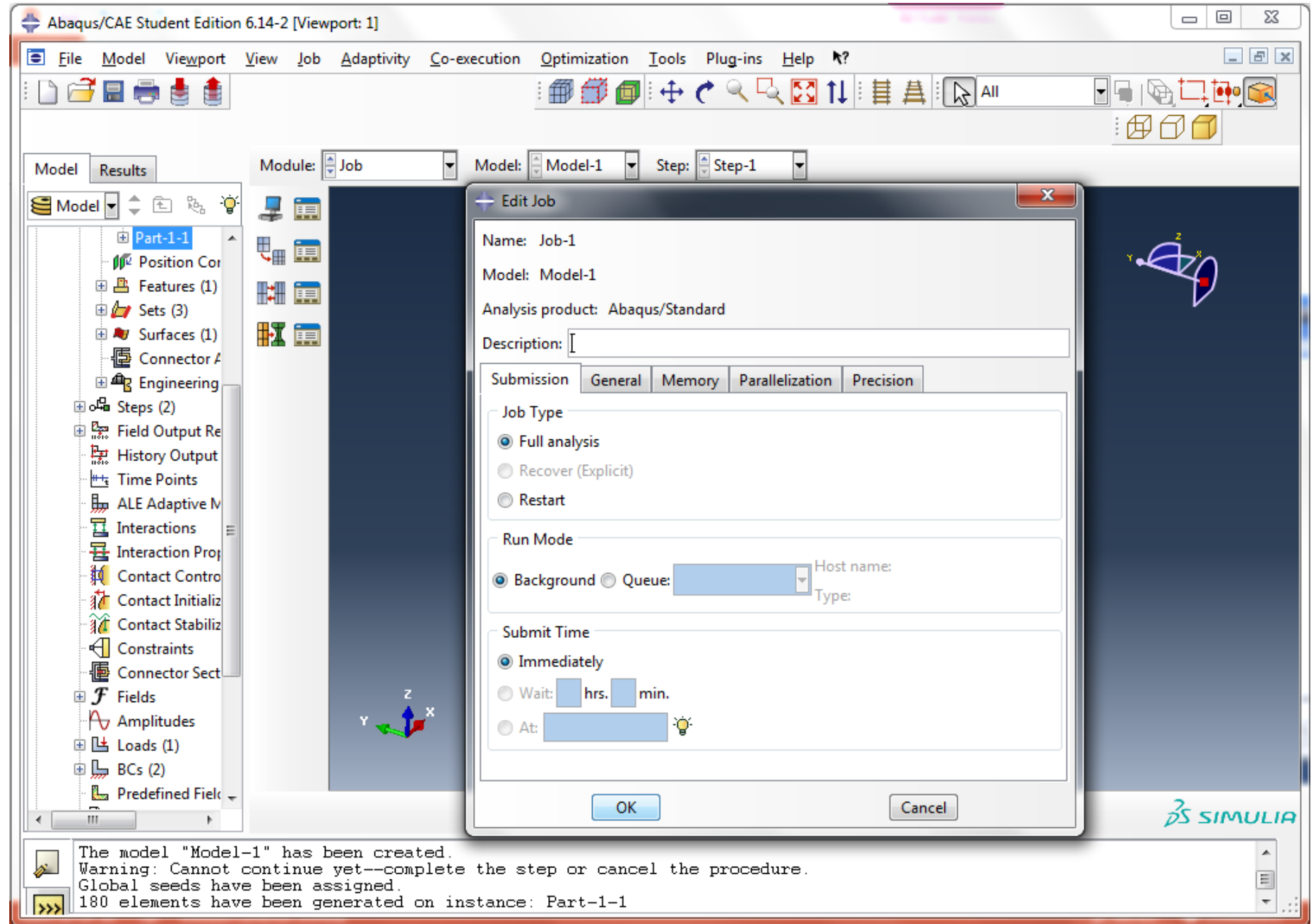


Click  the create the mesh

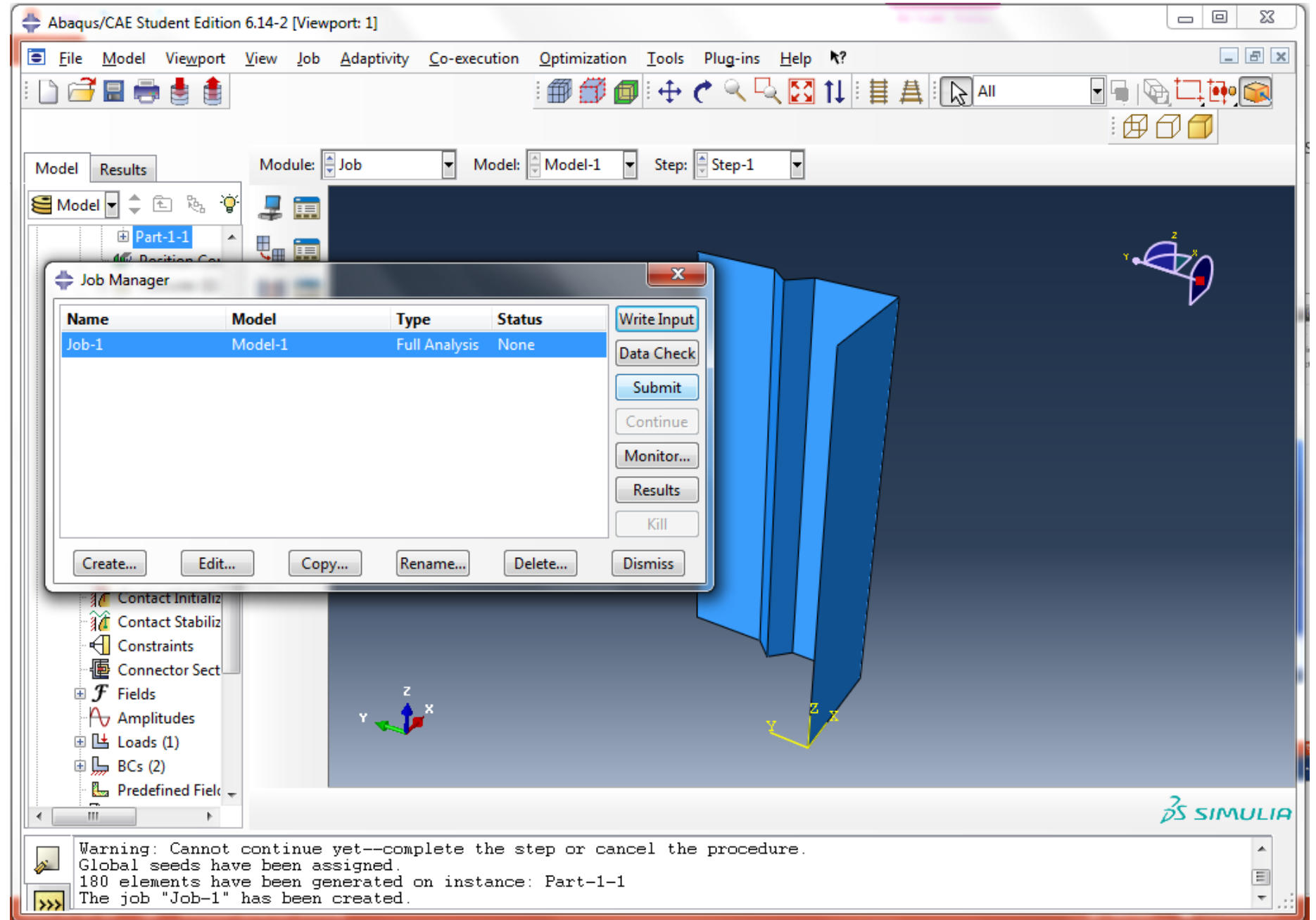
The default shell element type in ABAQUS is 4-node element which is fine for this analysis.



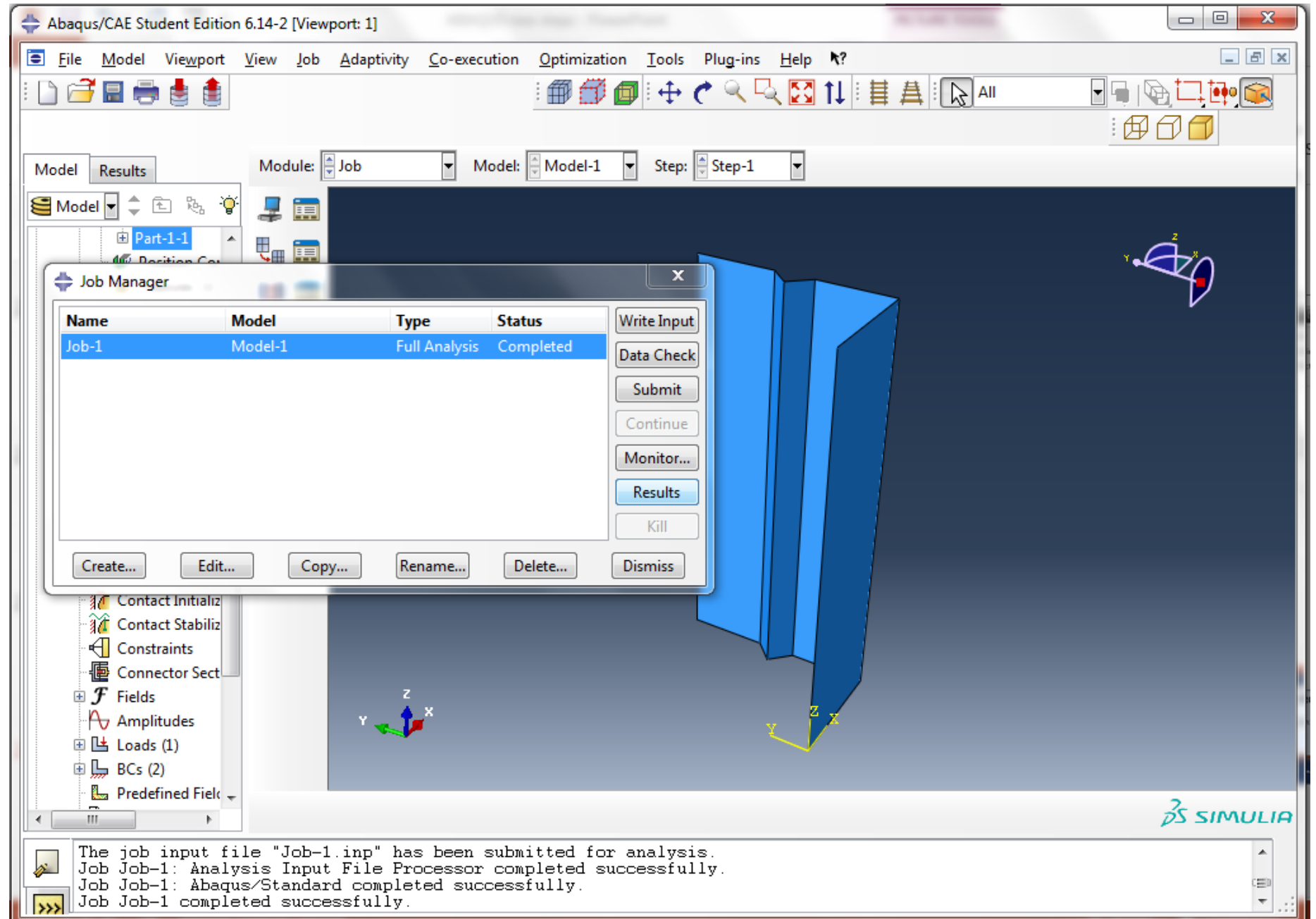
Use the default settings



“Submit” the job for analysis



Once the job is completed
Click "Results" to view the
results



We focus on the 1st mode.

The screenshot displays the Abaqus/CAE Student Edition 6.14-2 interface. The main viewport shows a 3D visualization of a structure's first mode of vibration. The displacement magnitude is color-coded, with a legend on the left indicating values from $-1.009e-00$ to $1.009e-00$. The structure is a vertical, slightly curved beam-like component. The interface includes a menu bar (File, Model, Viewport, View, Result, Plot, Animate, Report, Options, Tools, Plug-ins, Help), a toolbar with various icons, and a left-hand navigation pane with categories like Output Databases, Model Database, Spectrums, XYPlots, XYData, Paths, Display Groups, Free Body Cuts, Streams, Movies, and Images. The bottom status bar shows the following log messages:

```
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.
```

Under the “Result” tab, the “Step/Frame” option gives you the numerical results of the elastic buckling results

Use the absolute value of the 1st mode result as the critical elastic buckling force per unit length.

For this example, it is 1.1127 kips/in.

The screenshot shows the Abaqus/CAE Student Edition 6.14-2 interface. The main viewport displays a 3D visualization of a buckling mode, showing a vertical plate with a color gradient from blue to red, indicating the magnitude of the buckling displacement. The Step/Frame dialog box is open, showing the following data:

Step Name	Description
Step-1	

Index	Description
0	Increment 0: Base State
1	Mode 1: EigenValue = 1.1127
2	Mode 2: EigenValue = -1.1127
3	Mode 3: EigenValue = 2.6435
4	Mode 4: EigenValue = -2.6435
5	Mode 5: EigenValue = -6.0438

The dialog box has buttons for OK, Apply, Field Output..., and Cancel. The status bar at the bottom indicates that the analysis was completed successfully.