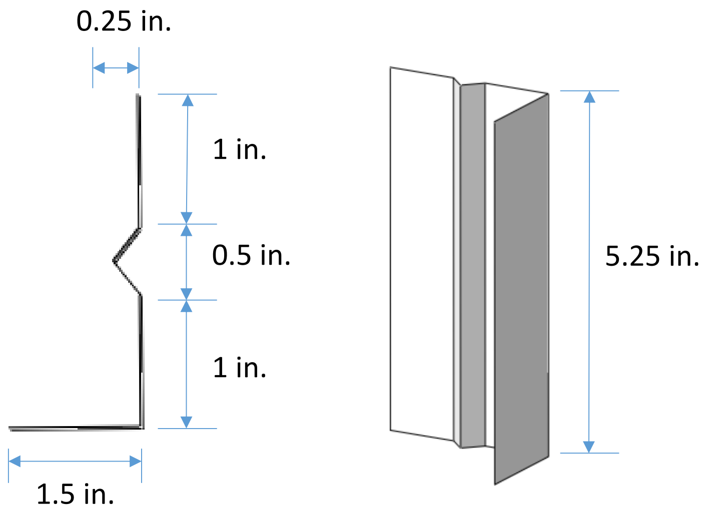
2015 International Student Competition on Cold-Formed Steel Design

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.

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

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

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

$V_n = minimim\left(0.17\lambda^{-0.8}F_yBt, \qquad 0.35F_yBt\right)$

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

$= minmum\,(1.182\ kips, 2.735\ kips)$

$= 1.182\ kips = 1182\ lbf$

The total weight:

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

Nominal shear strength per unit weight:

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

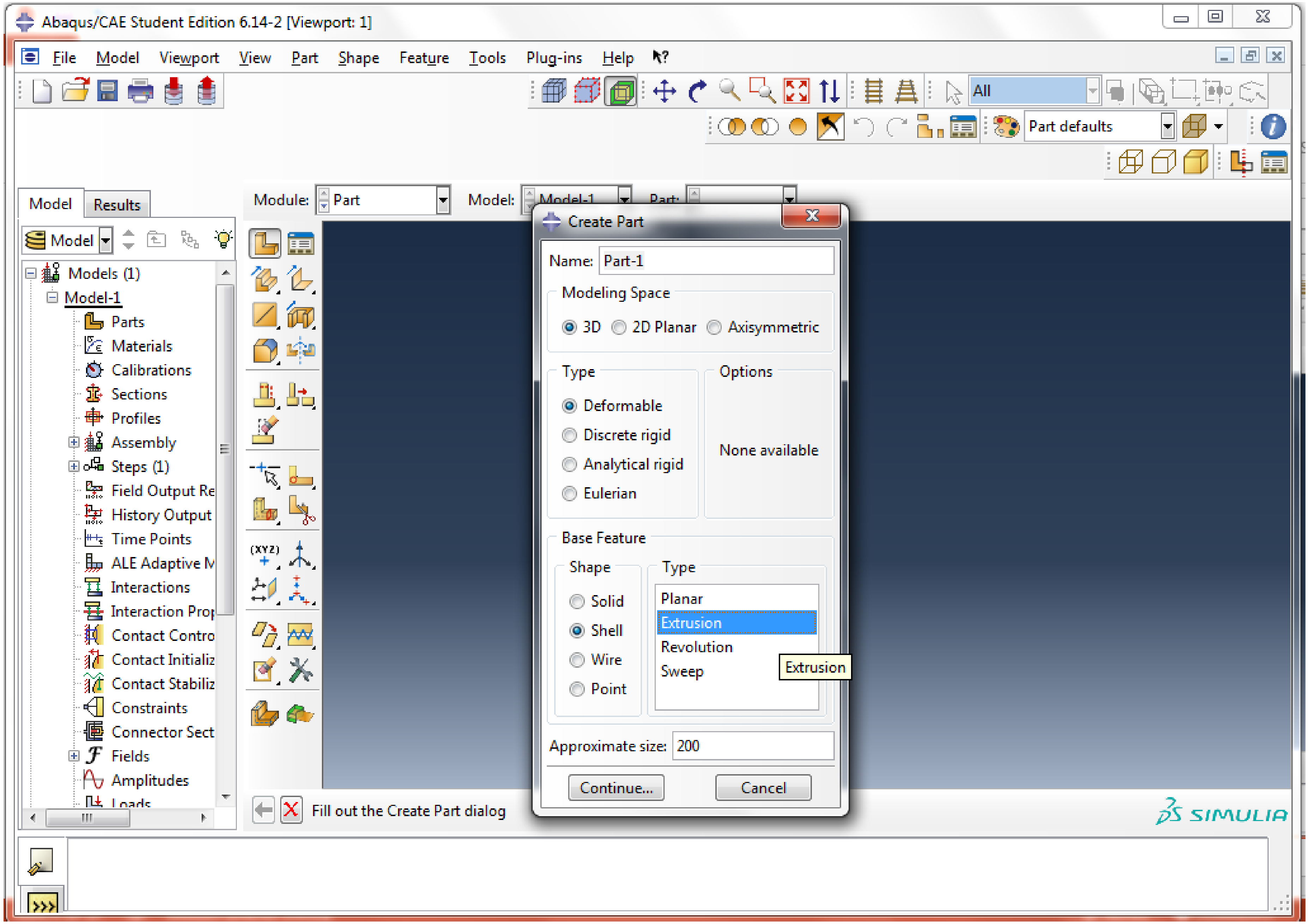
The following pages include the screen shots of the step-by-step procedure of the ABAQUS modeling and analysis. The ABAQUS student edition can be obtained from 3DS Academy website http://academy.3ds.com/software/simulia/abaqus-student-edition/

Step 1: create part

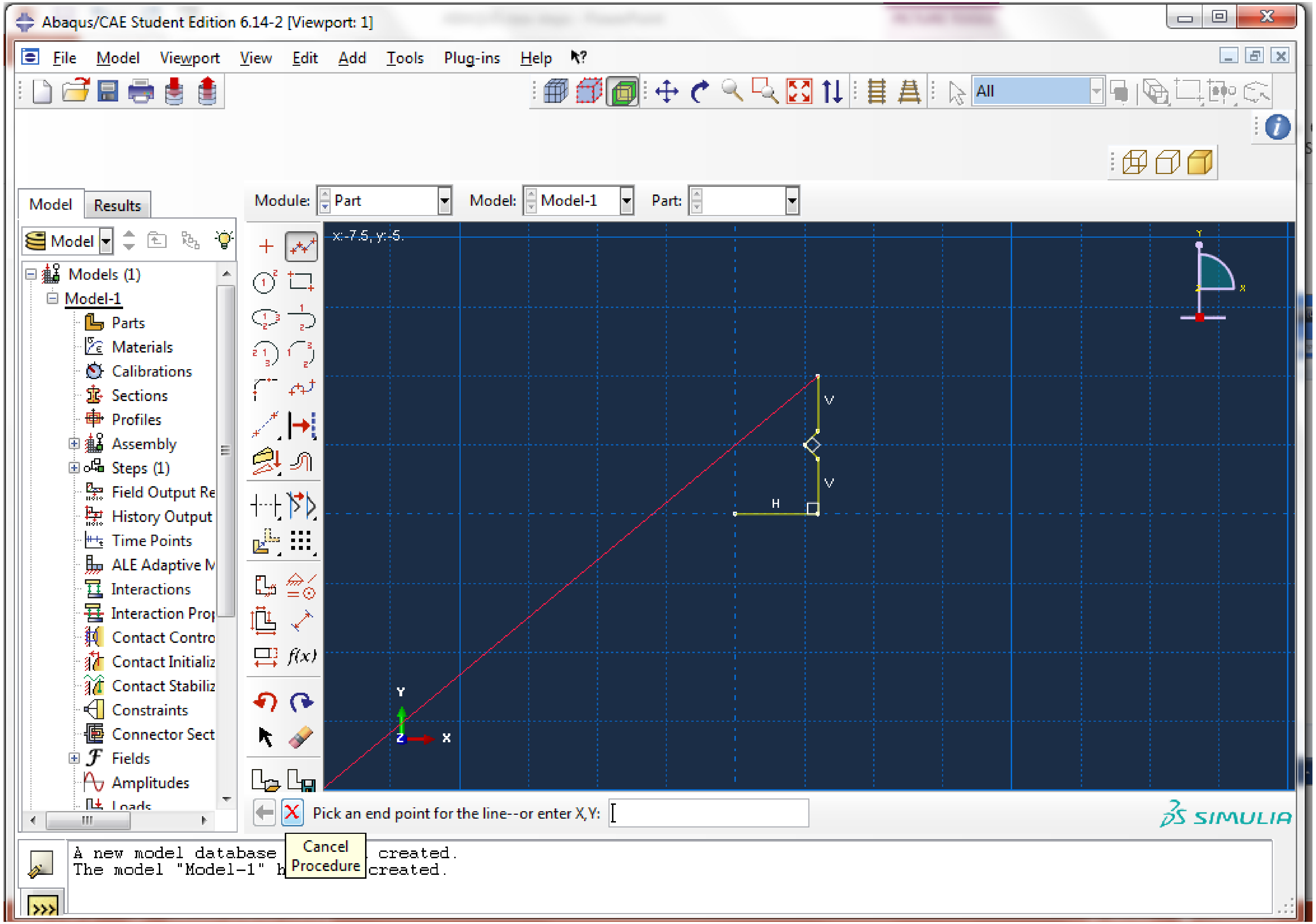
We will use shell elements for the clip angle model. We create the 3D model using extrusion from a shell shape.

Start with the “Part” Modulus

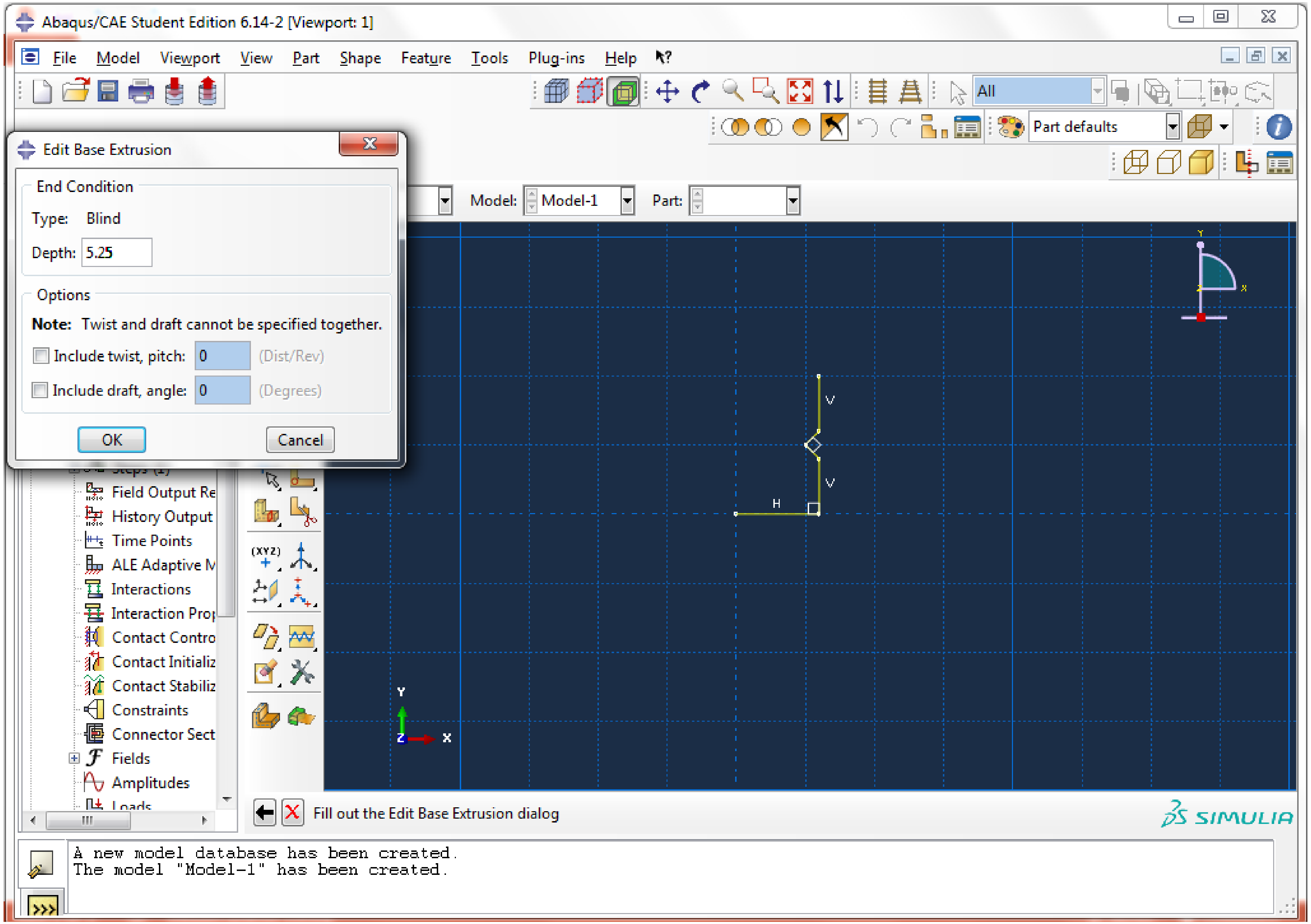
Click  to create part



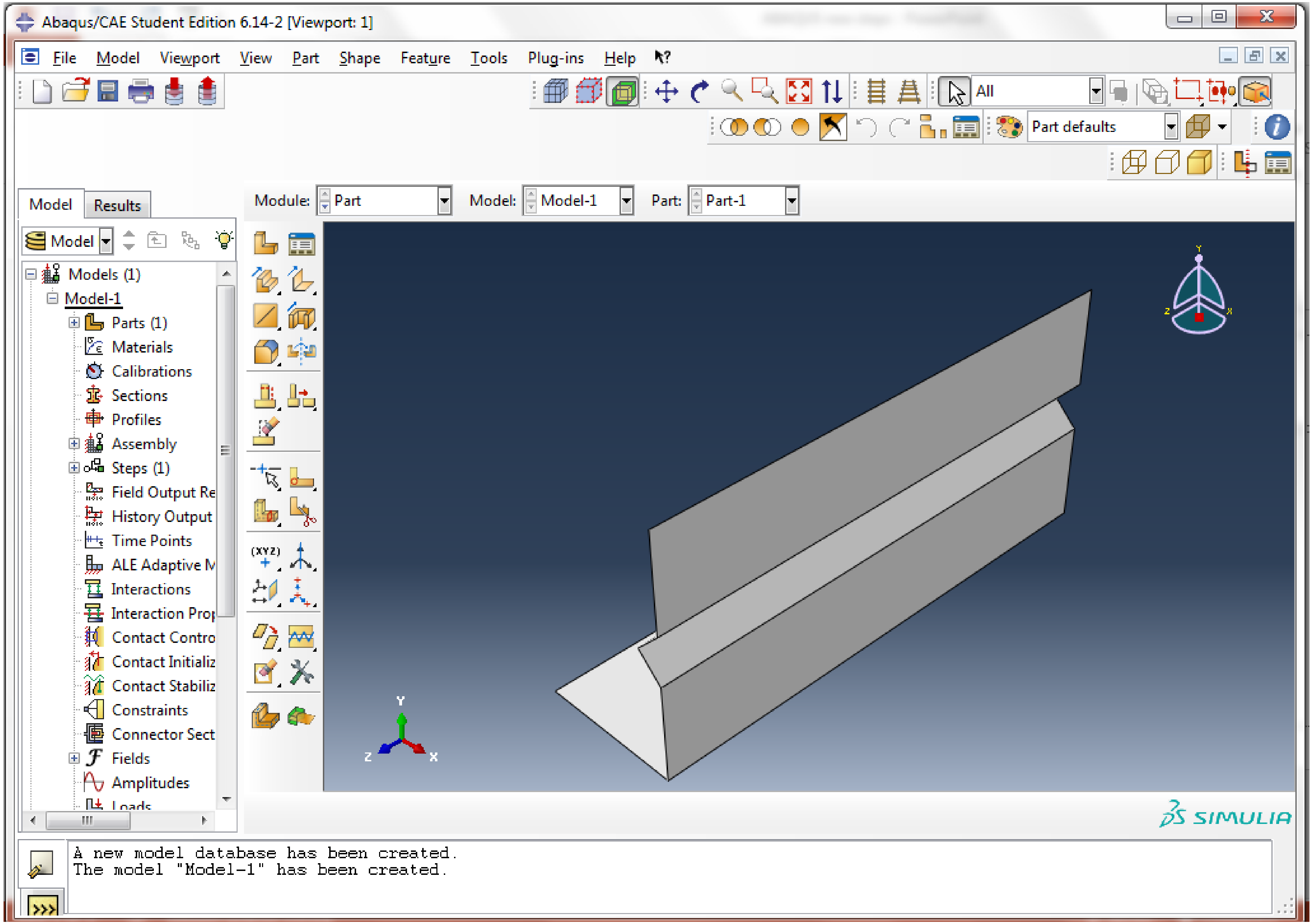
Click to create the cross section shape.
Click to finish.



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



Step 1 finished.

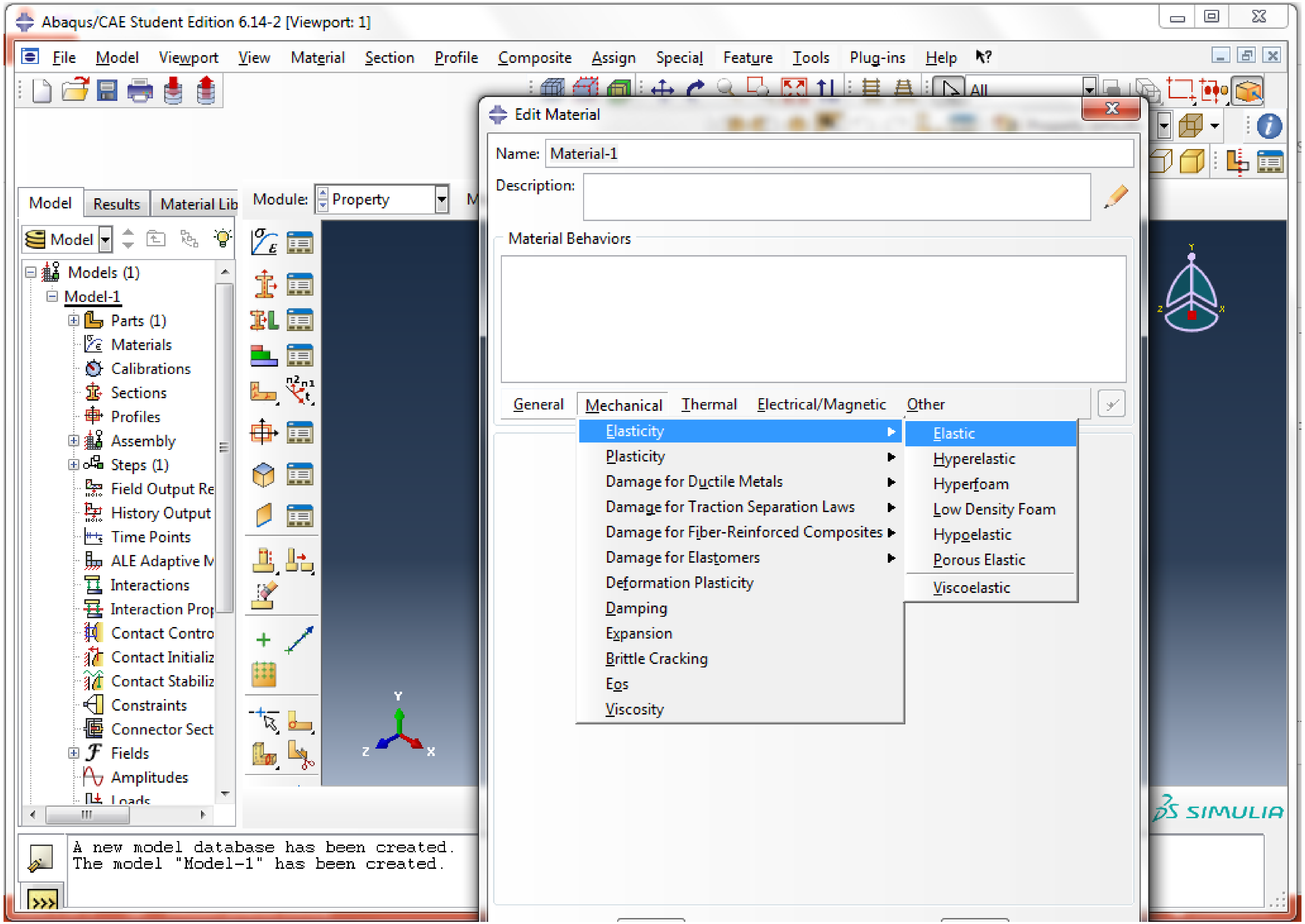


Step 2 create material model and section

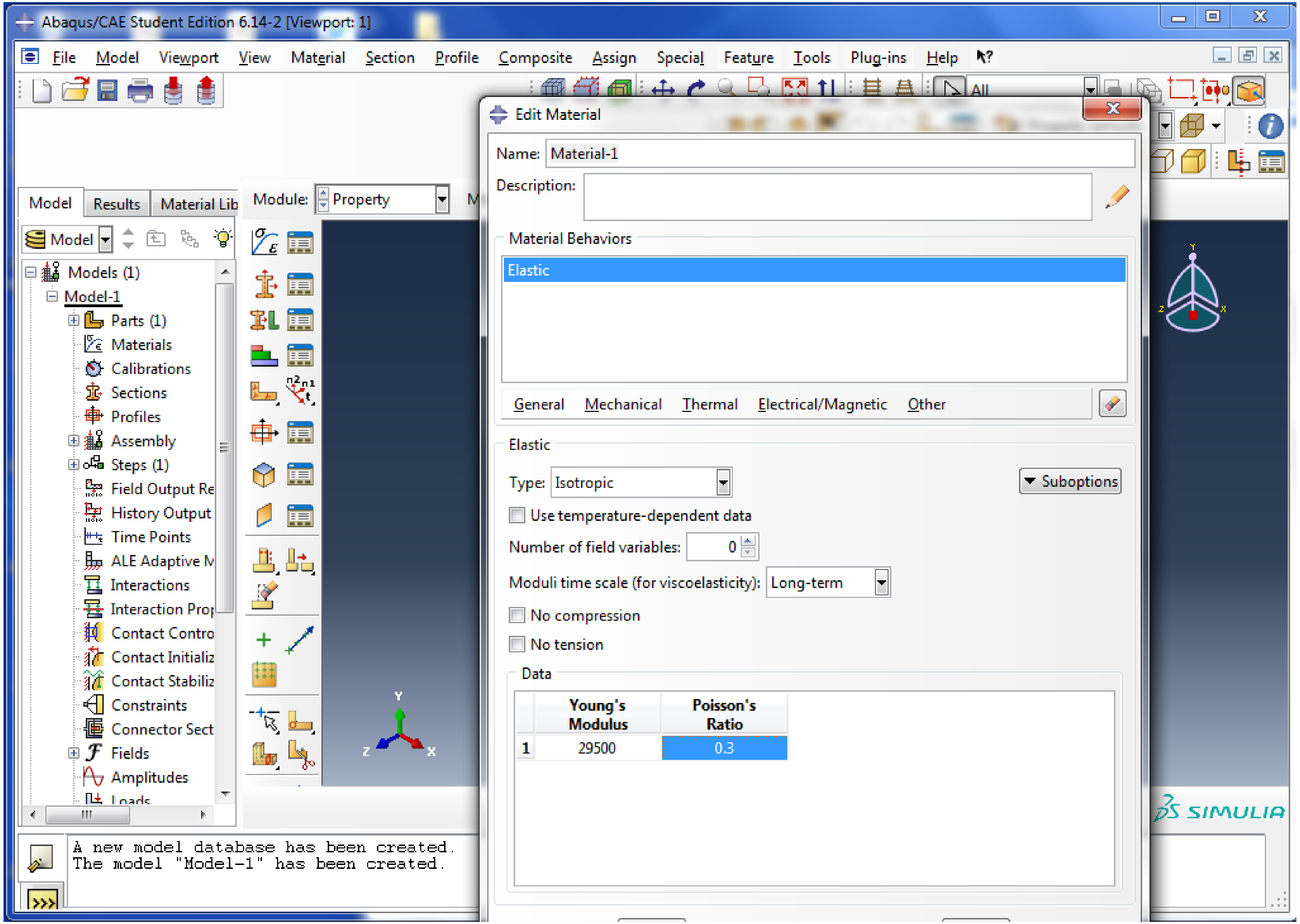
Choose the “Property” Modulus

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

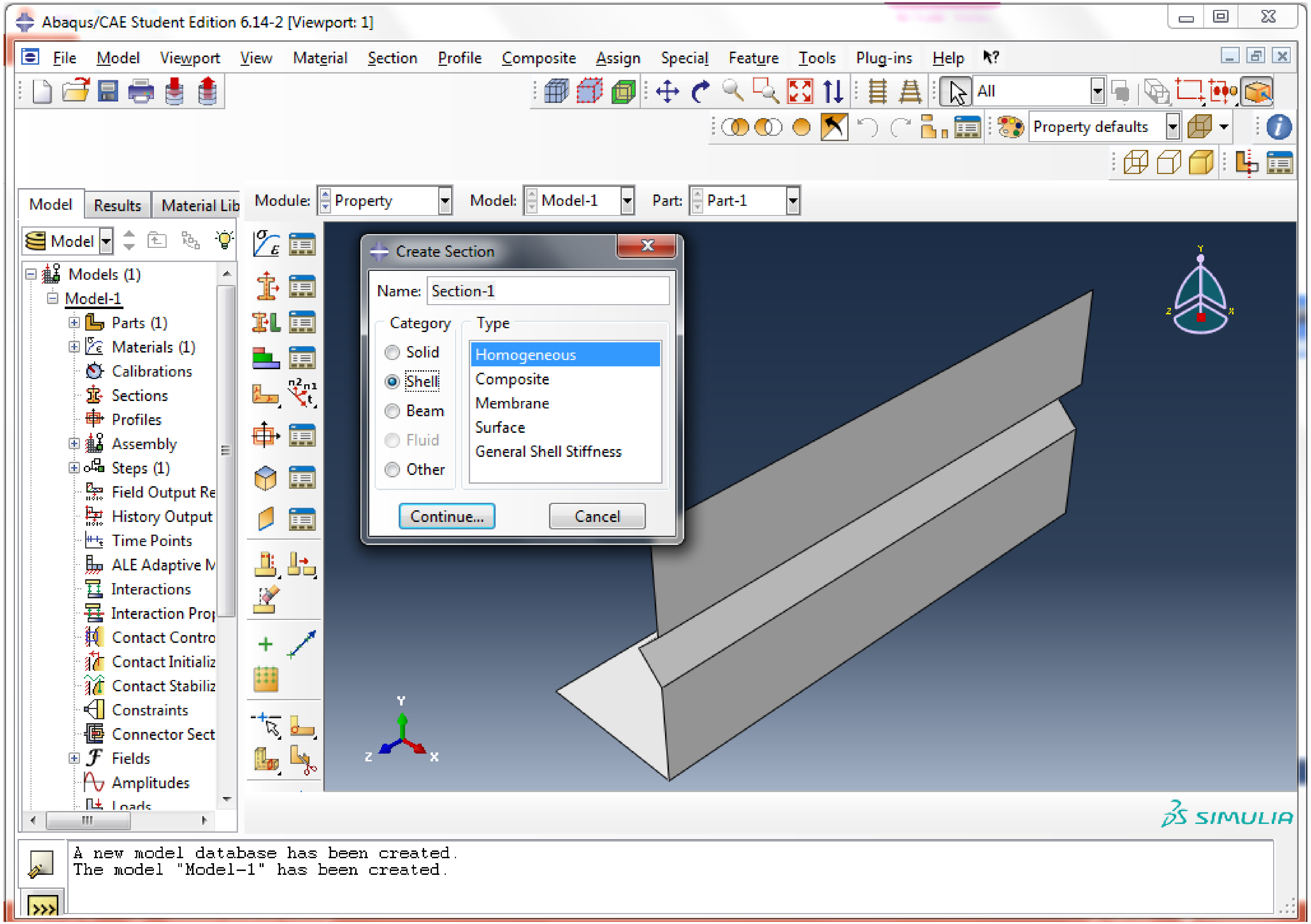
We are doing an elastic buckling analysis here.



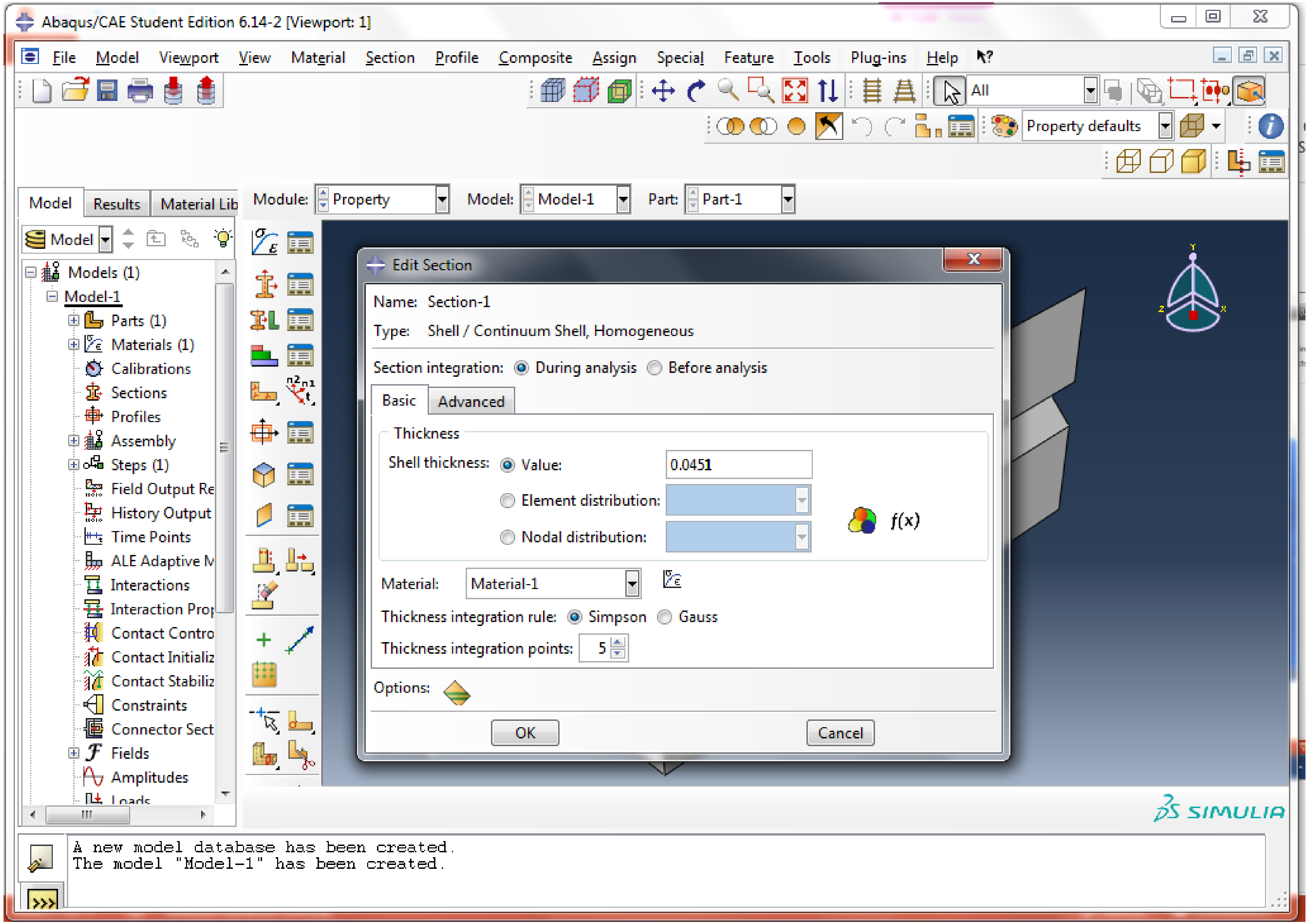
Input Young's modulus, 29500 (ksi);
Possion's ration 0.3



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



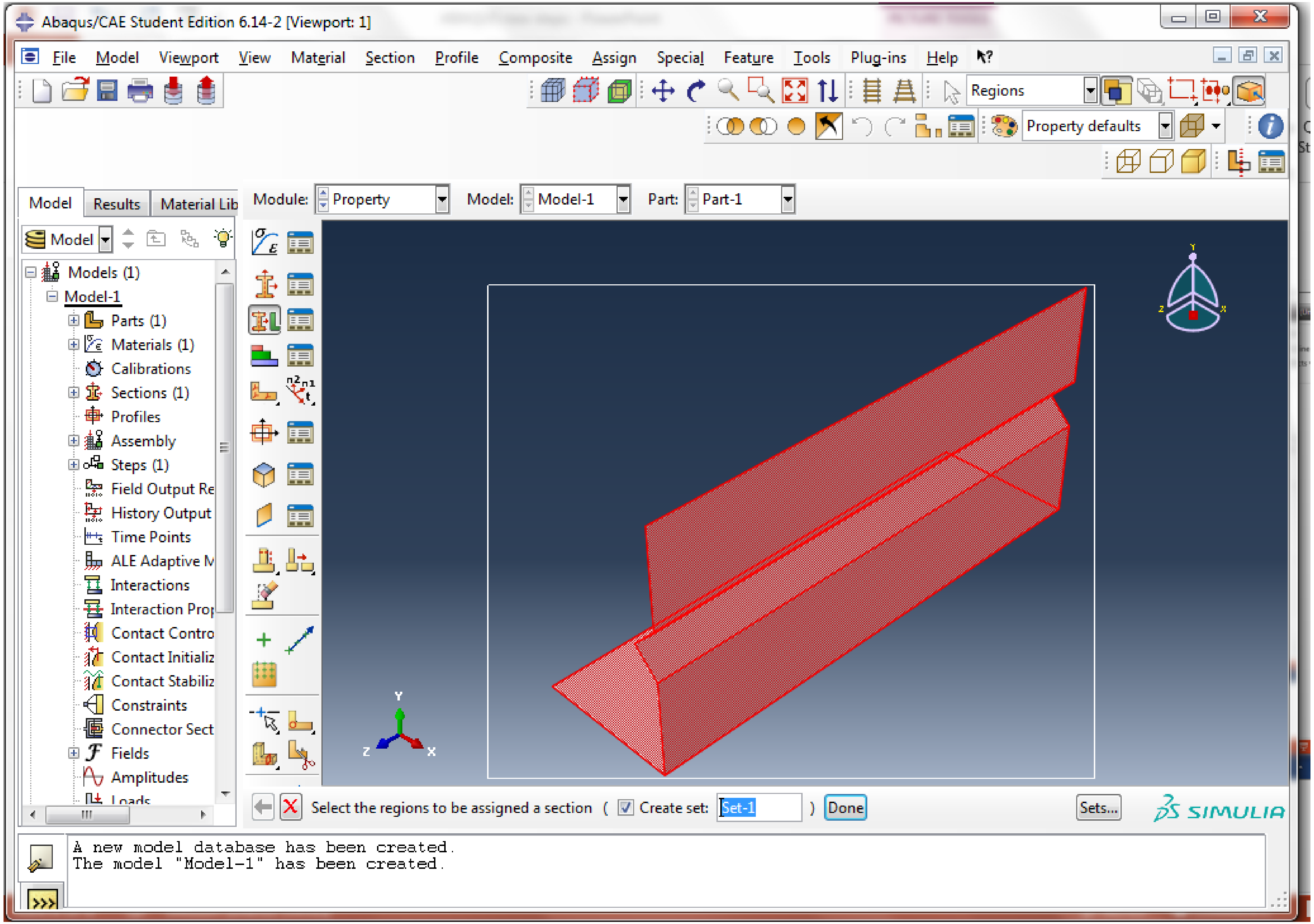
We define the thickness, 0.0451 (in.) in the section property



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

Select the entire clip angle.

Step 2 finished.

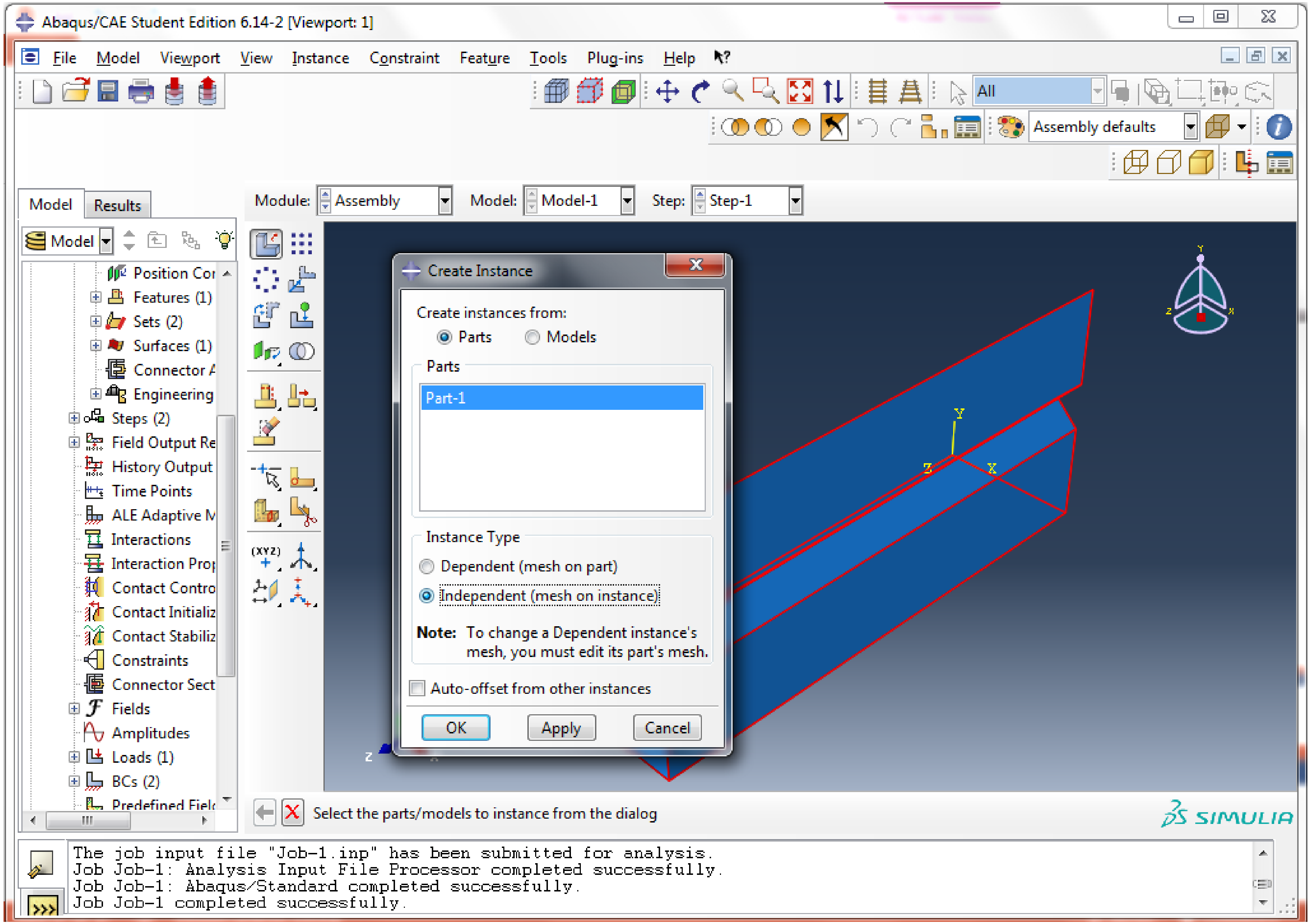


Step 3 create instance

In the “Assembly” Module

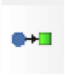
Click  to create an instance using the “Independent (mesh on instance)” Type.

Step 3 finished.

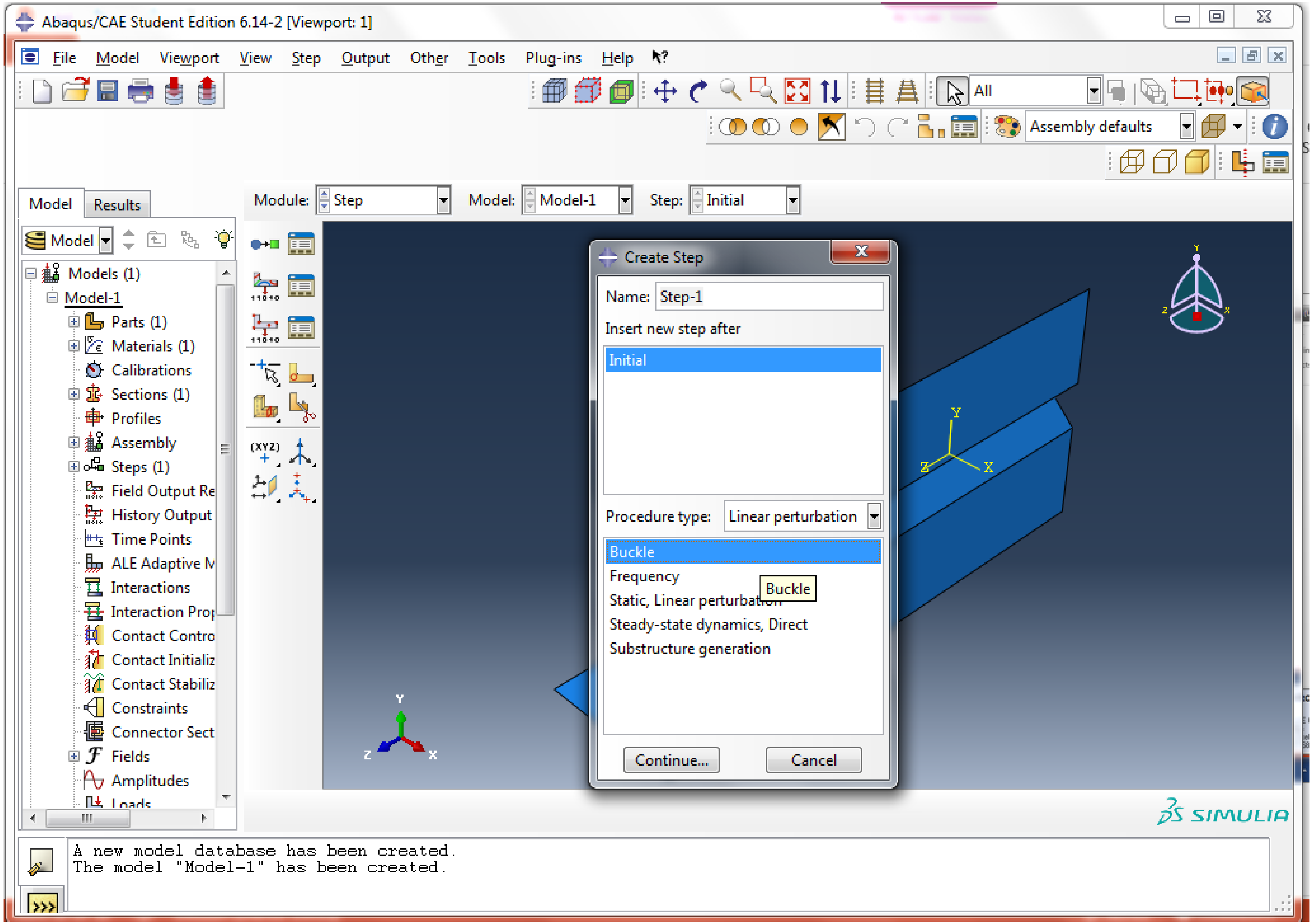


Step 4 create steps

In the “Step” Module

Click  to create step

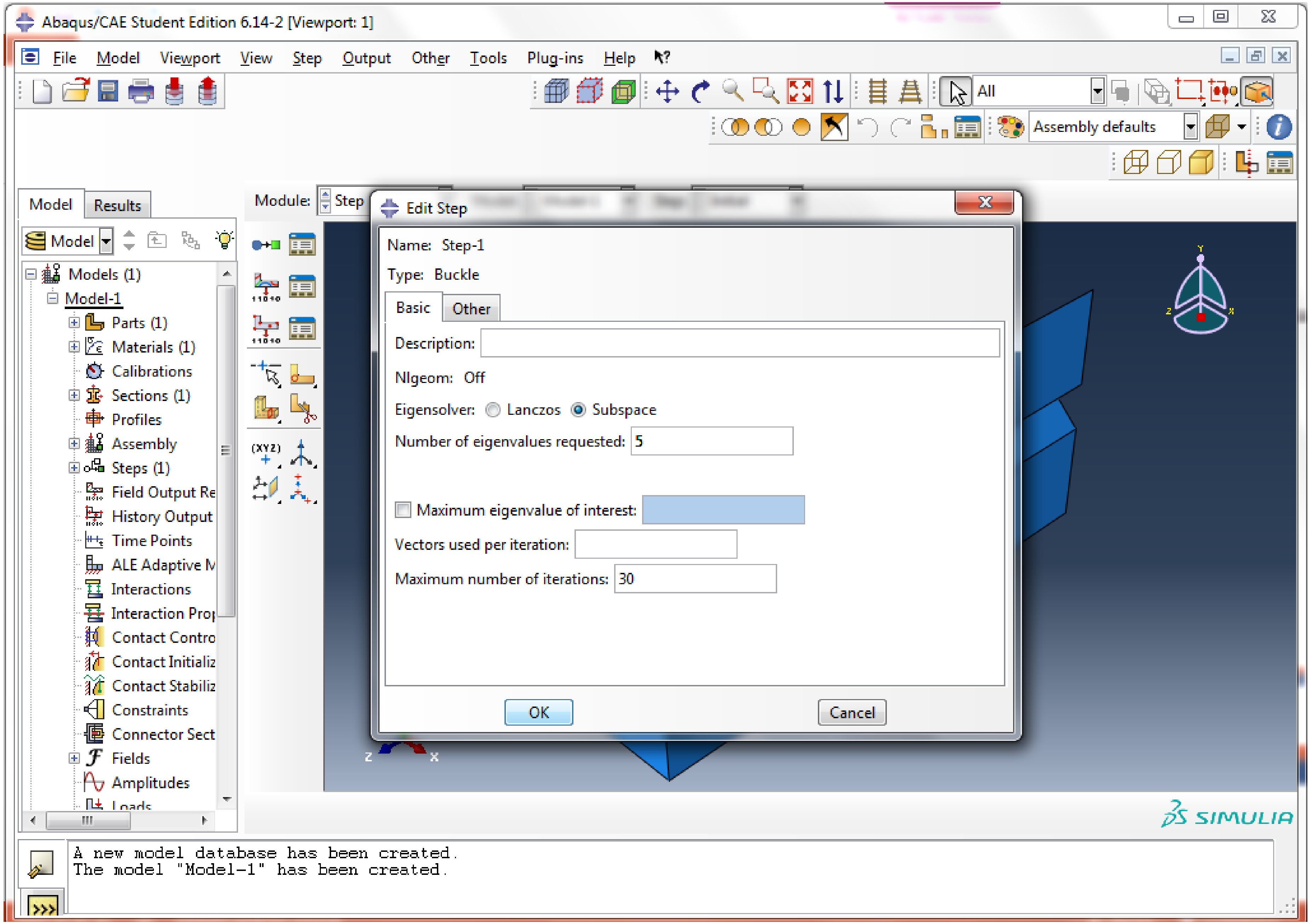
Since we are doing an elastic buckling/eigenvalue analysis, we use “Linear perturbation” as Procedure type and “Buckle” as the analysis type.



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



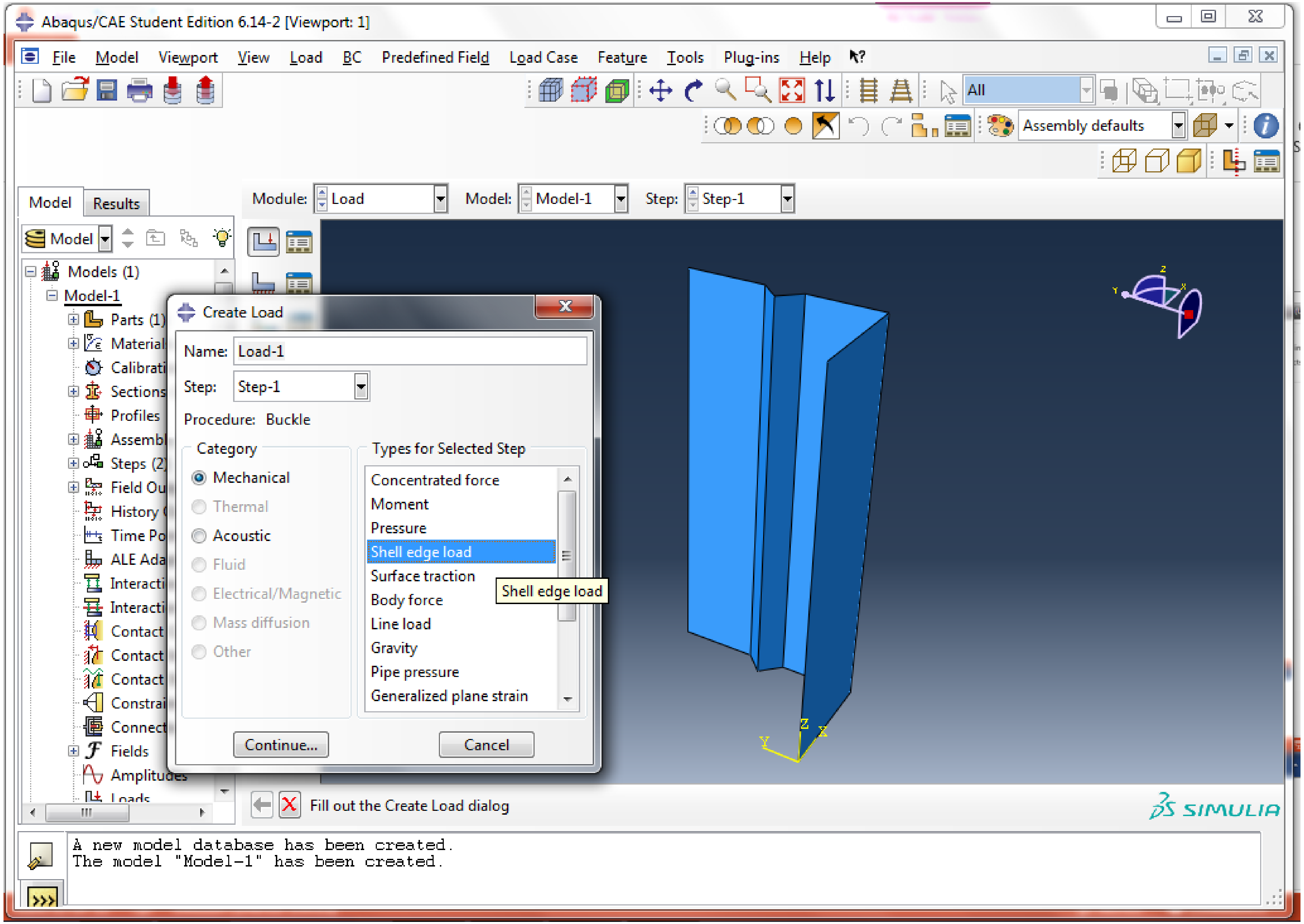
Step 5 create load and boundary conditions

In the “Load” module

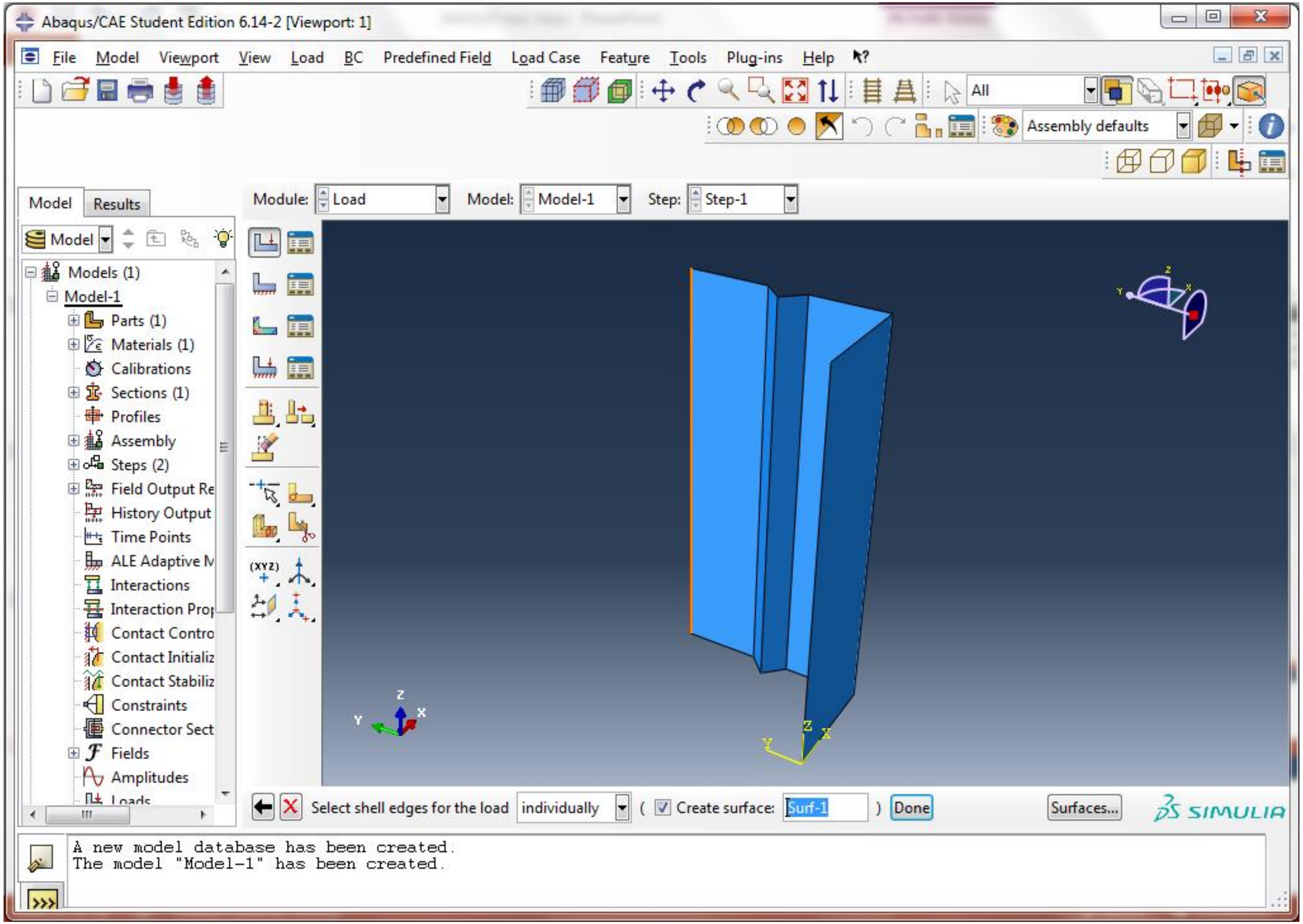
Click to create the uniform shear force on the edge of cantilevered leg.

We use “Shell edge load.”

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

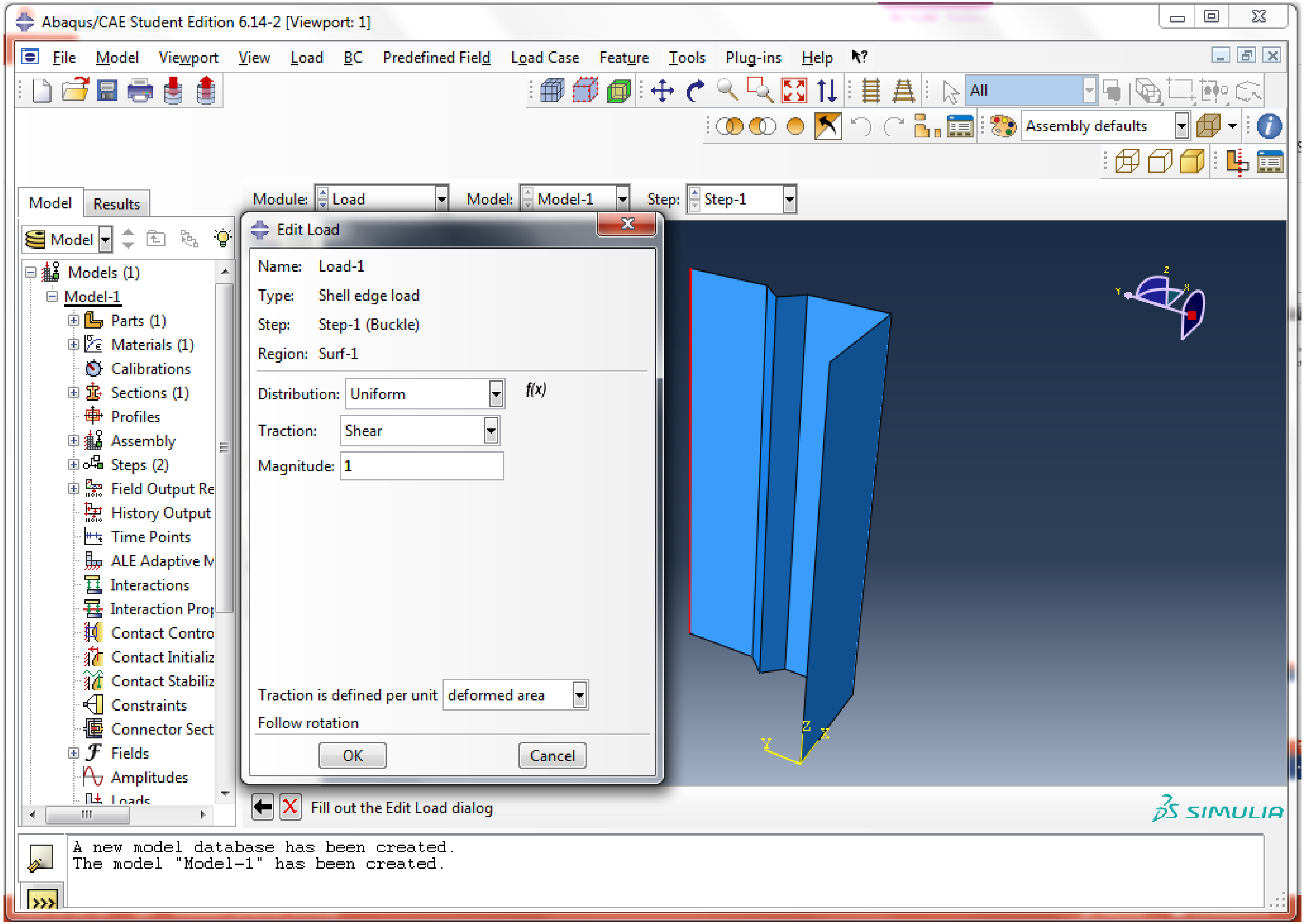


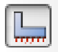
Select the edge of the cantilevered leg for the applied shear load

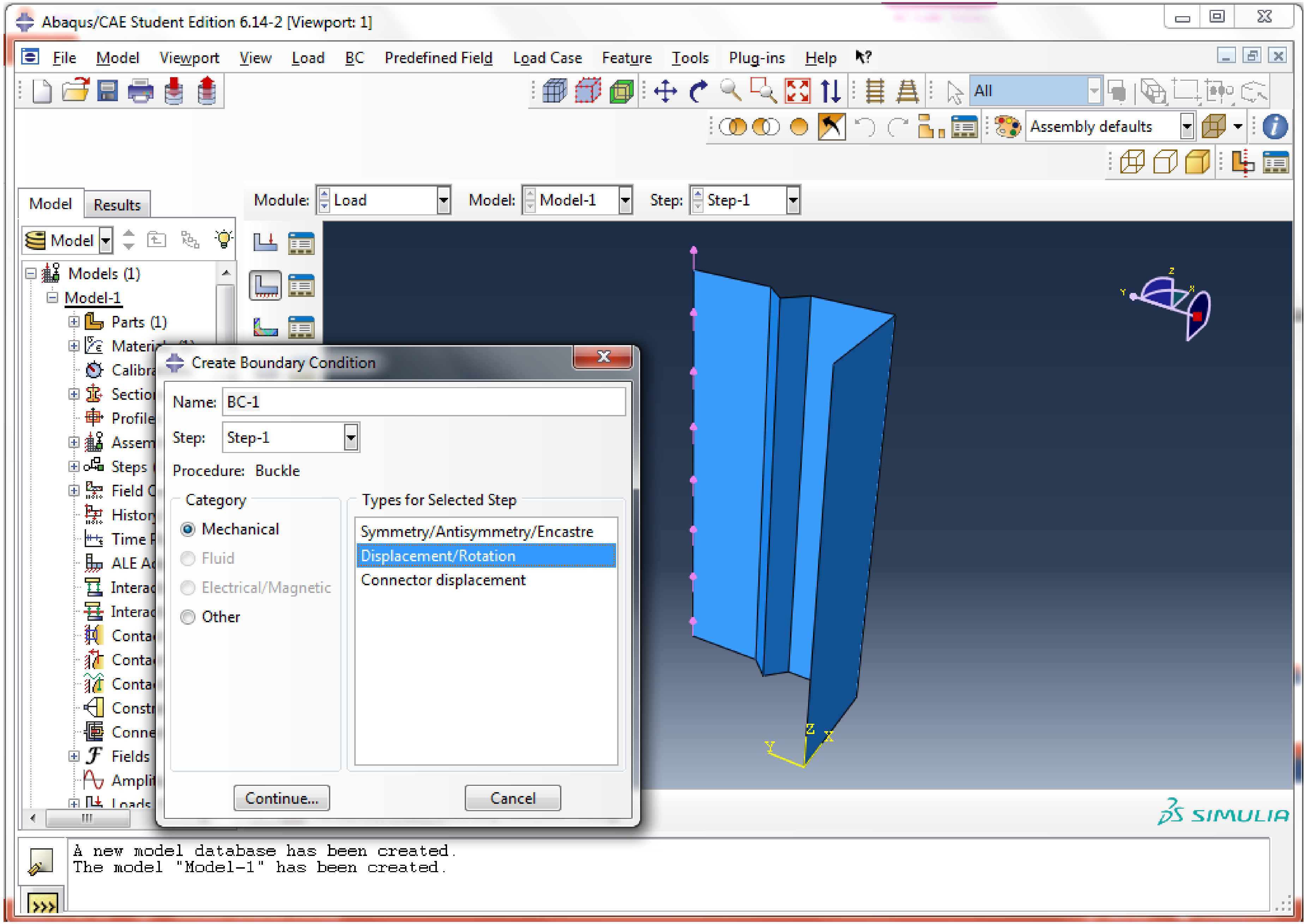


Specify the shell edge load:

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

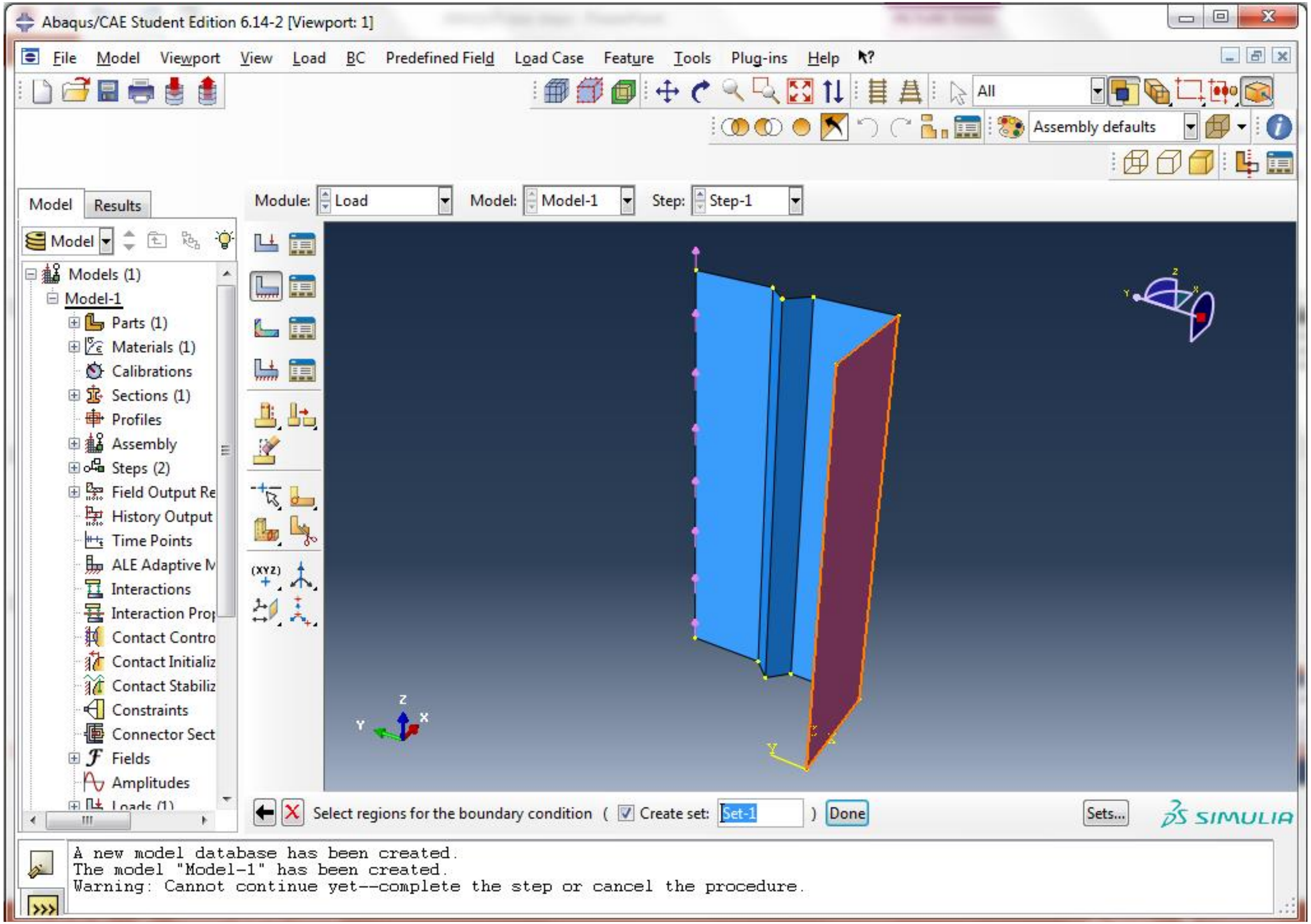


Click  to define boundary conditions. We use "Displacement/Rotation" type.



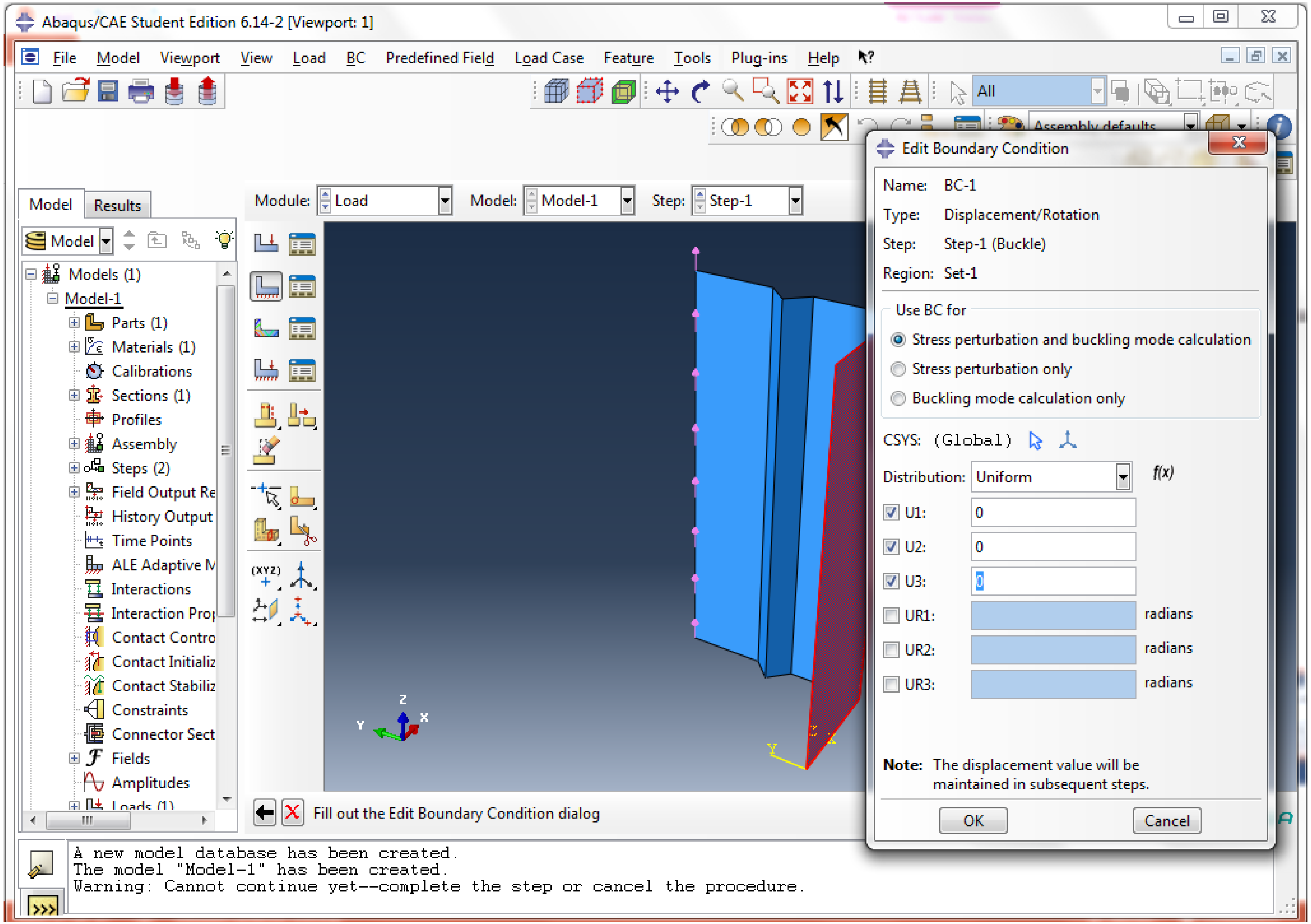
The first boundary condition is the fixed surface of the anchored leg.

Select the bottom surface of the anchored leg.



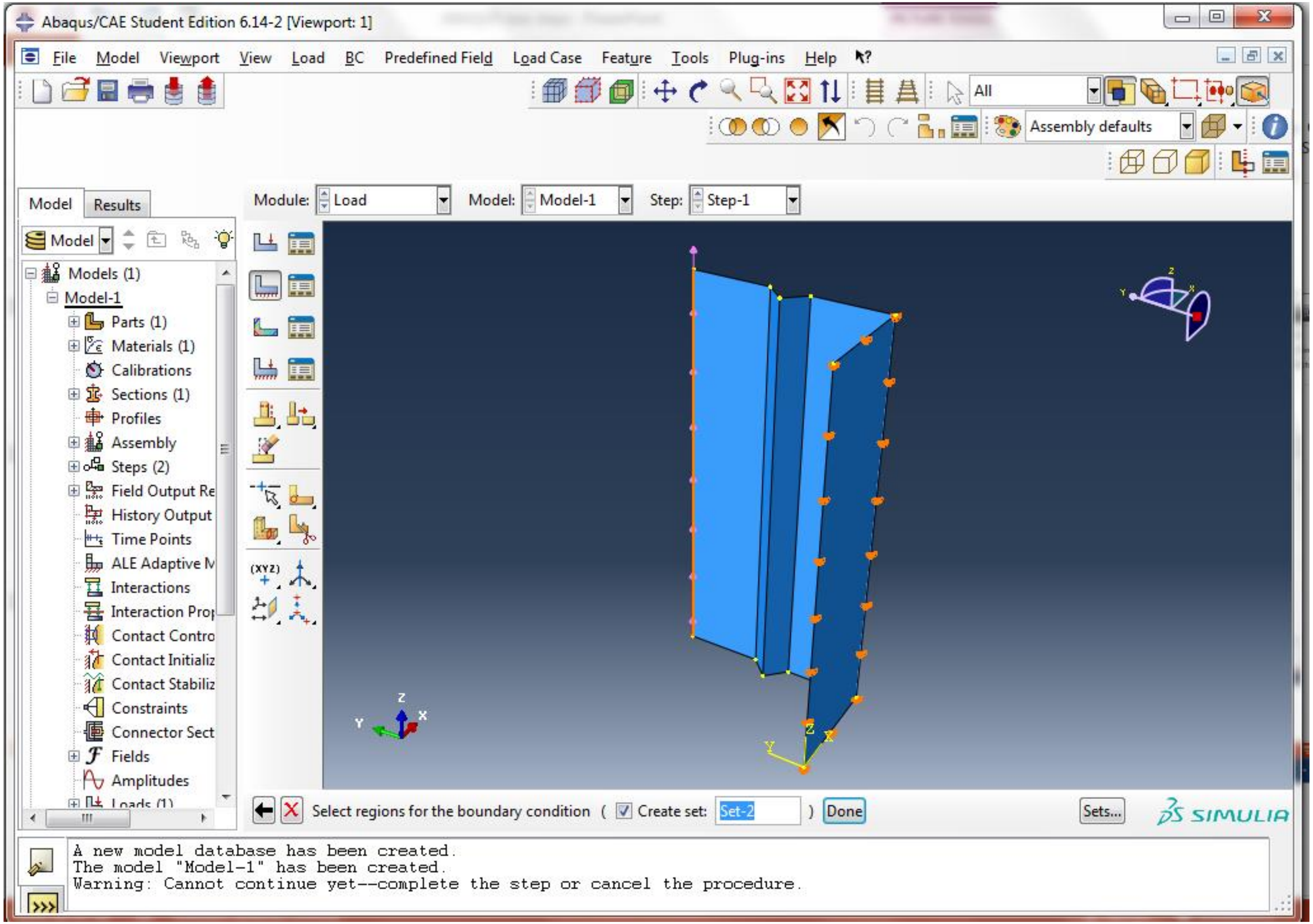
We fix the displacement of the surface in all 3 directions.

U1=0
U2=0
U3=0



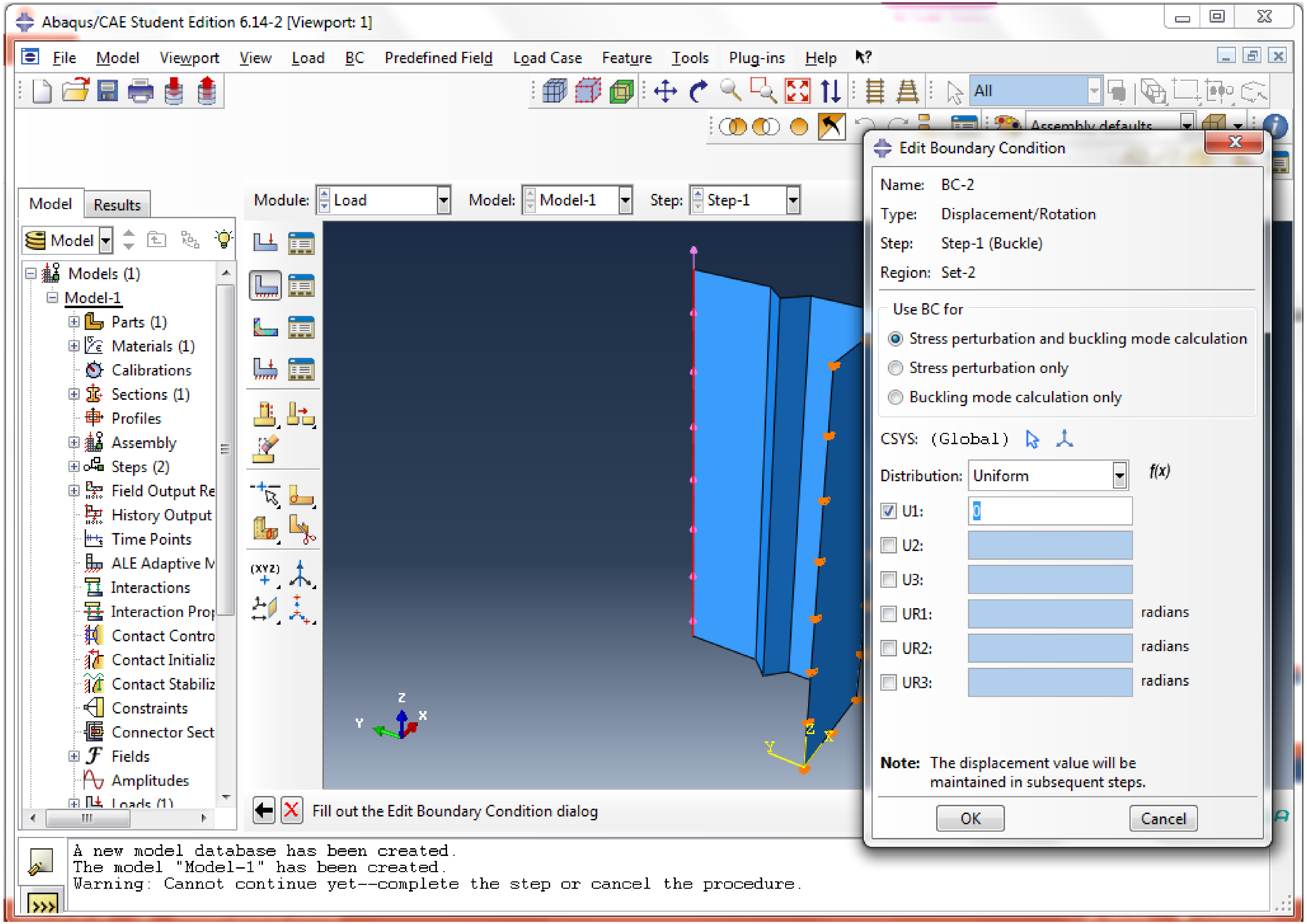
The 2nd boundary condition (BC) is the lateral constraint on the loaded edge.

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



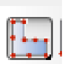
For the BC2, we only fix the out-of-plane movement U1=0

Step 5 completed.

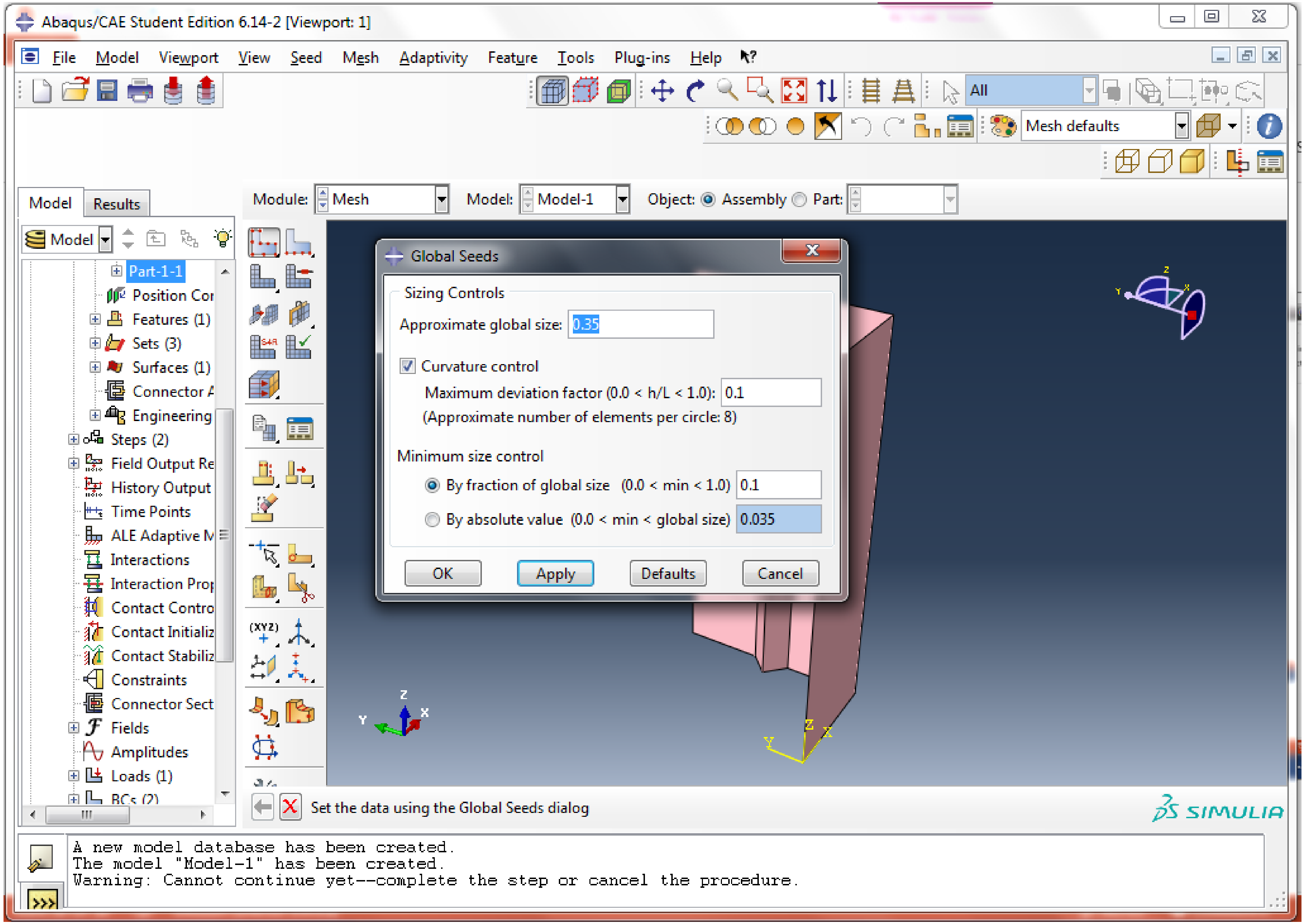


Step 6 create mesh

In the “Mesh” Module

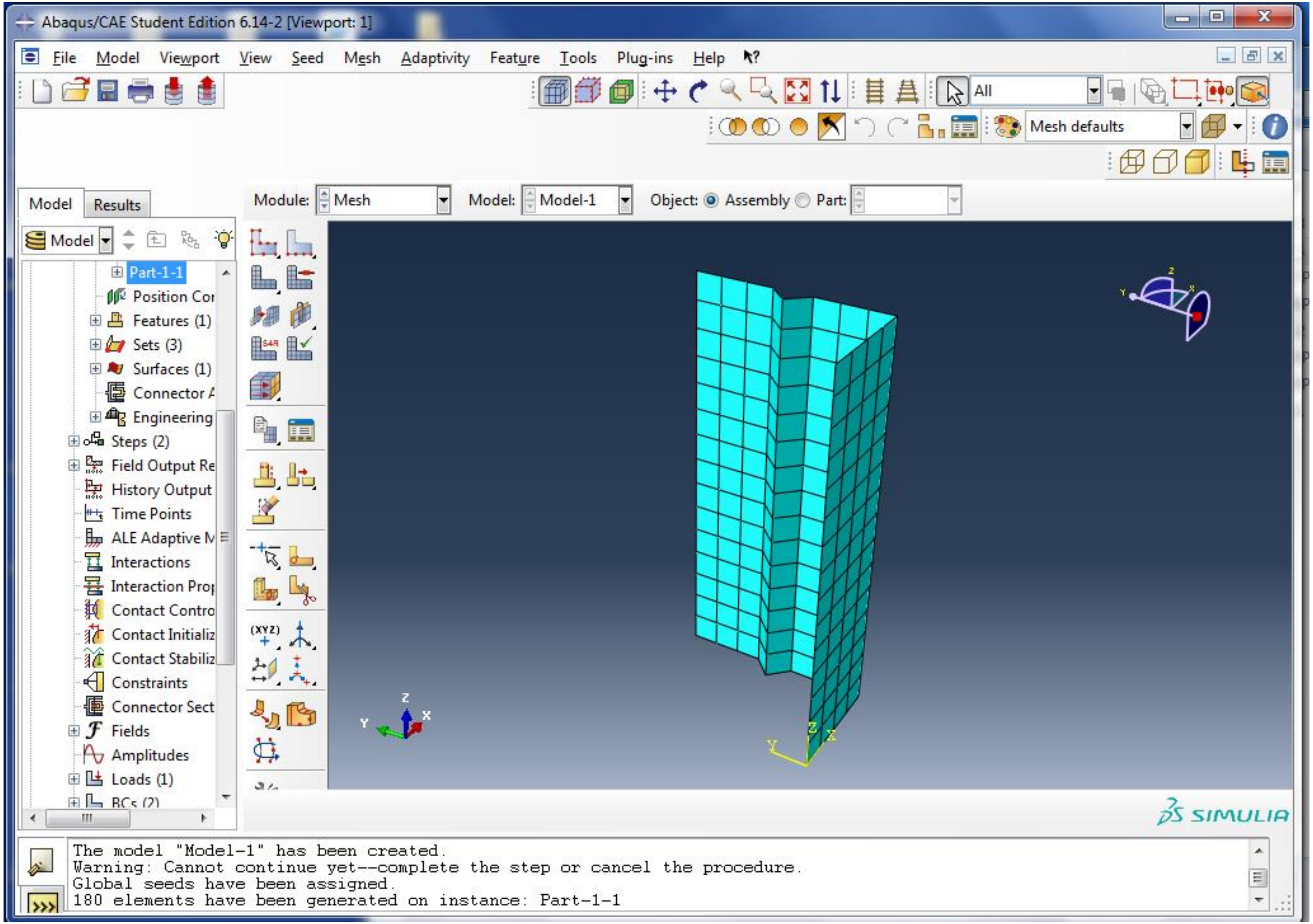
Click  to define the mesh sizes.

You can use the default setting.

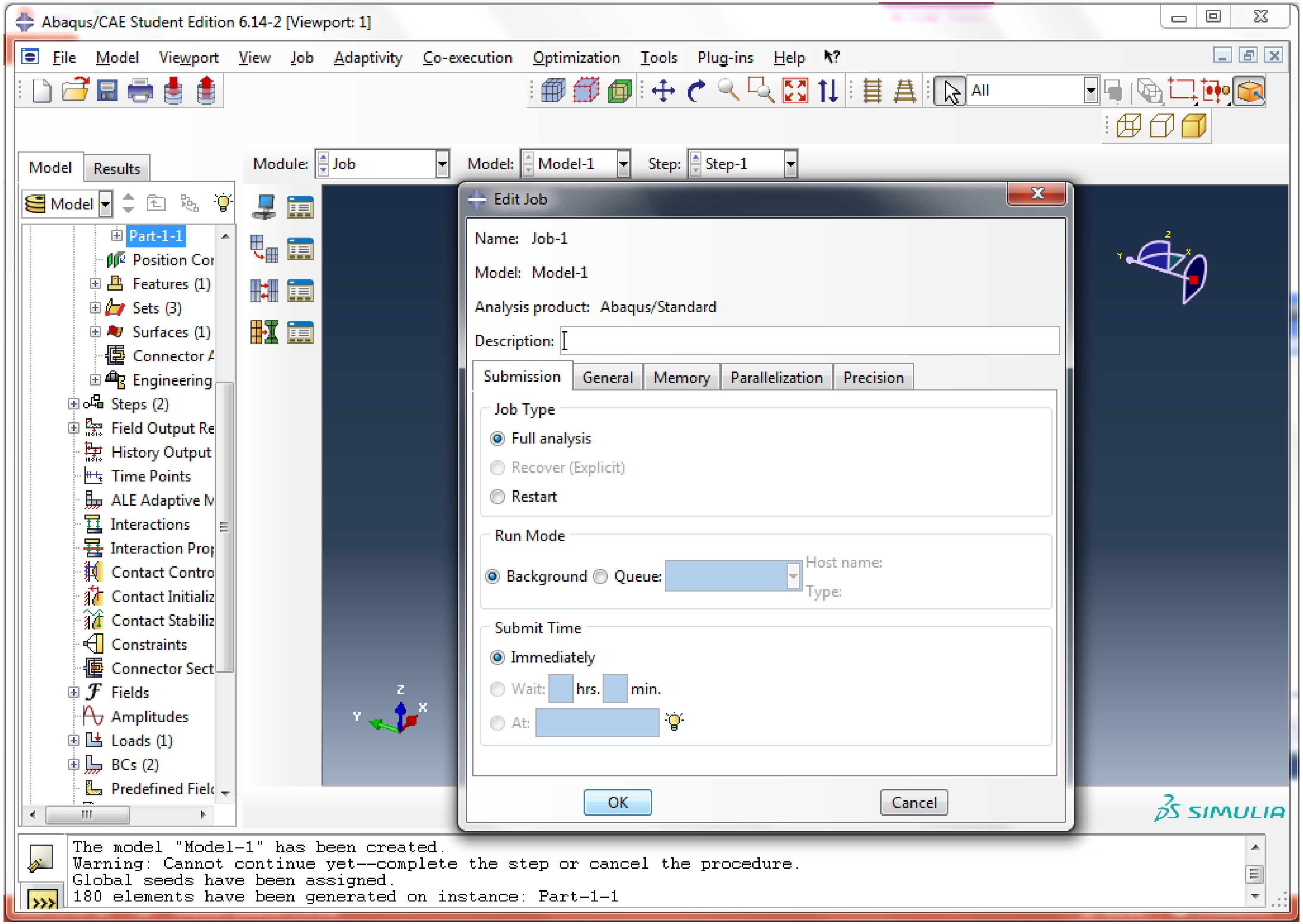


Click the create the mesh

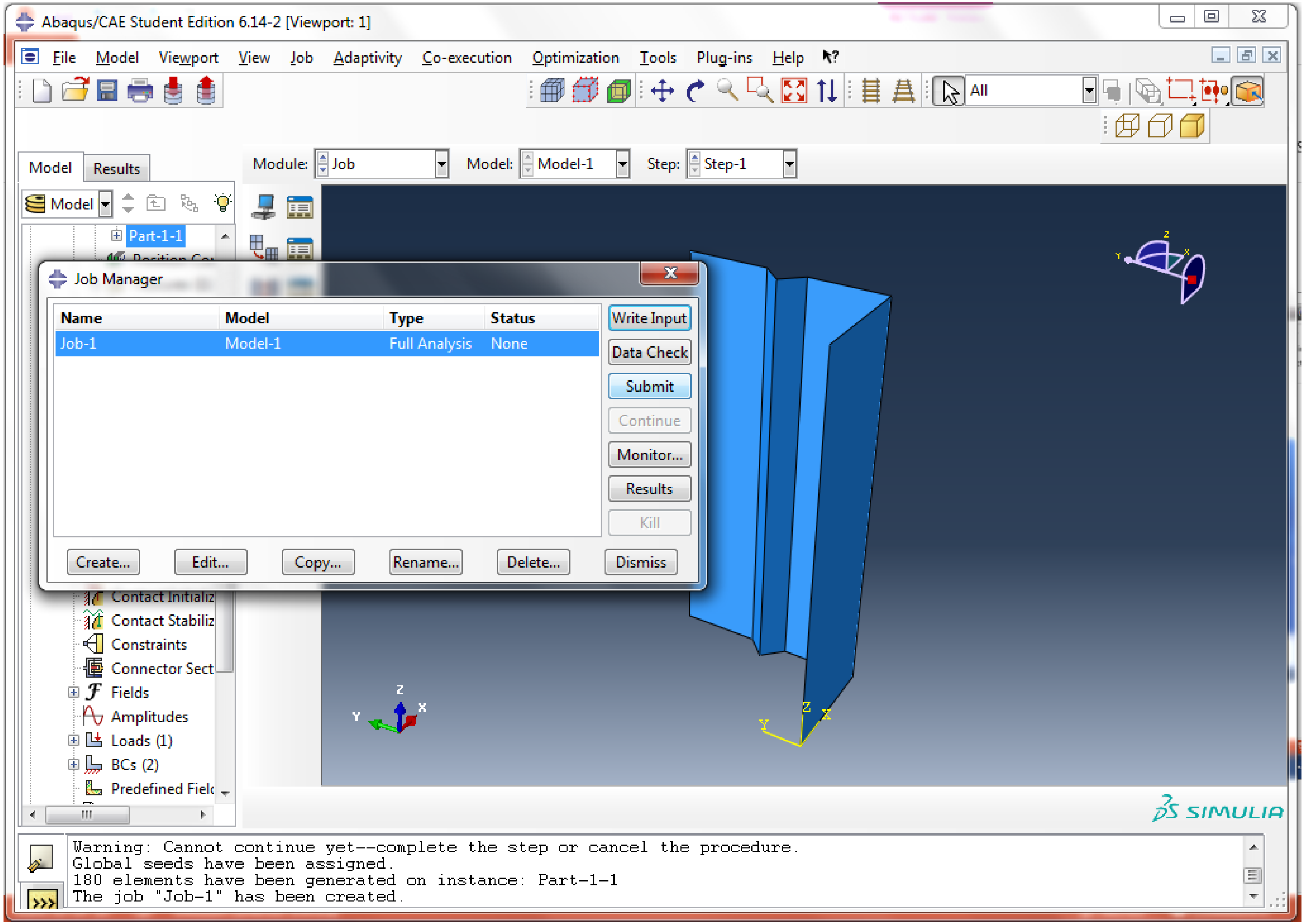
The default shell element type in ABAQUS is 4-node element which is fine for this analysis.



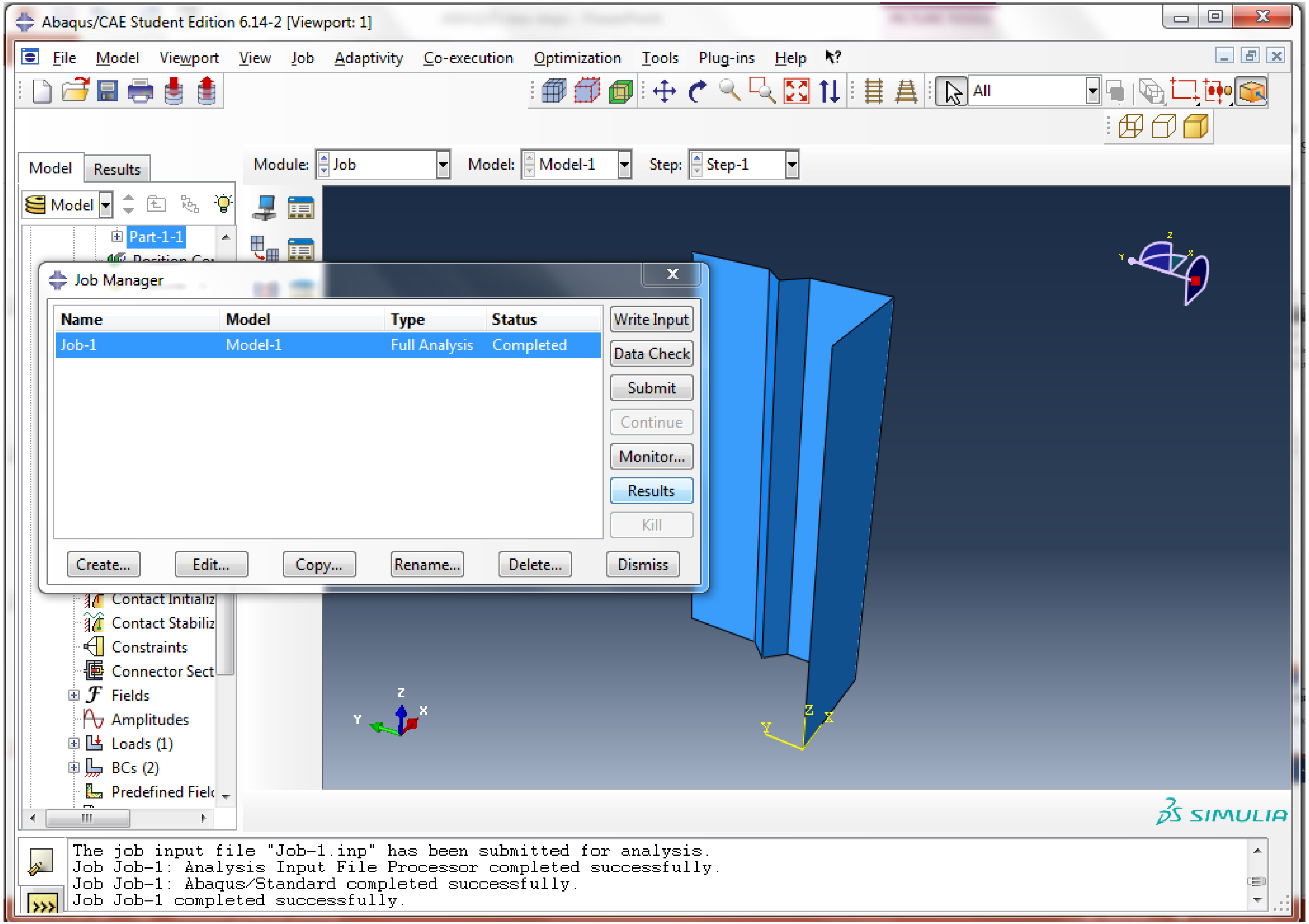
Use the default settings



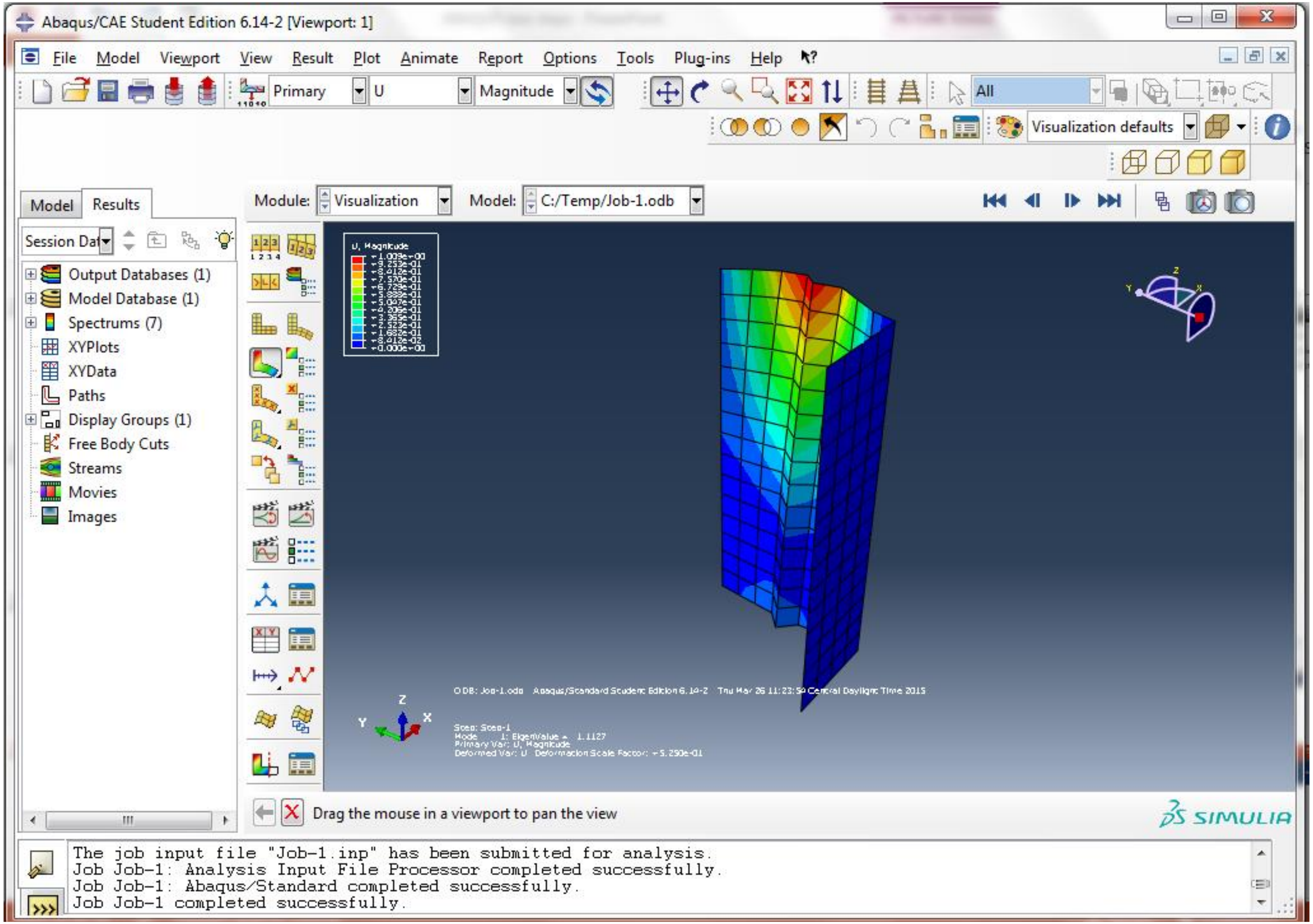
“Submit” the job for analysis



Once the job is completed Click “Results” to view the results



We focus on the 1st mode.



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

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

For this example, it is 1.1127 kips/in.