

Module *Part*


Click the **Create Part** icon in the **Toolbox** area.

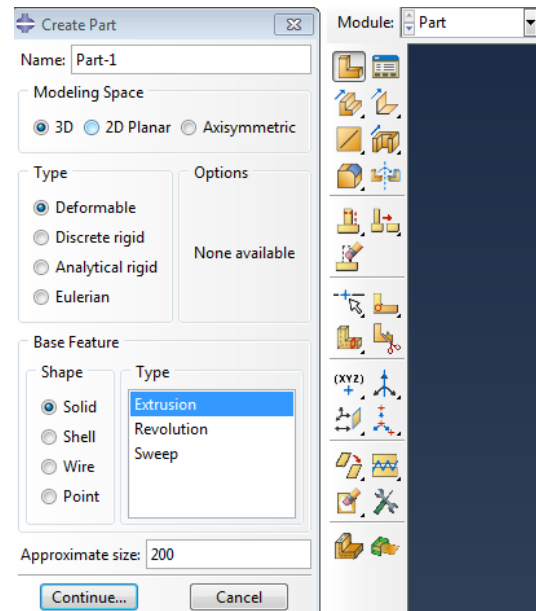
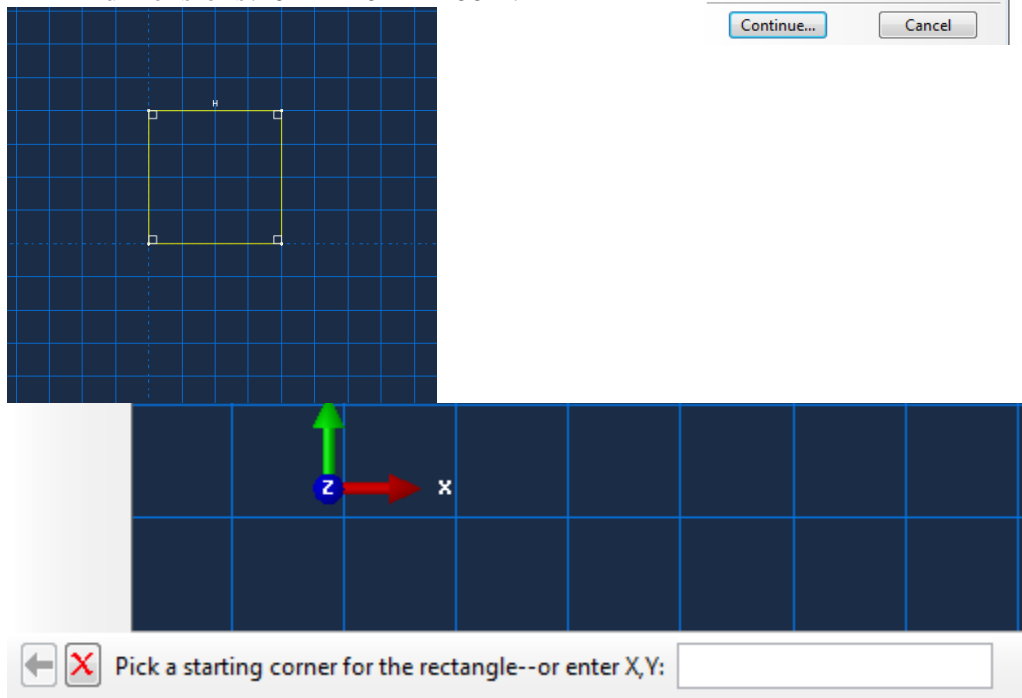
The **Create Part** dialog box appears.

1. Name the part **Beam** and select a 3D deformable body with a solid base feature.
2. For **Approximate size**, type **200**.

Click **Continue** to exit the **Create Part** dialog box.

Now ABAQUS/CAE automatically enters **Sketcher**.

1. Select the **Create Lines:Rectangle** tool , located in the upper-right region of the **Sketcher** toolbox.
2. Click **Done** in the **Prompt** area to exit the **Sketcher**.
3. Create a sketch according to the following dimensions: $20\text{m} \times 20\text{m} \times 200\text{m}$.



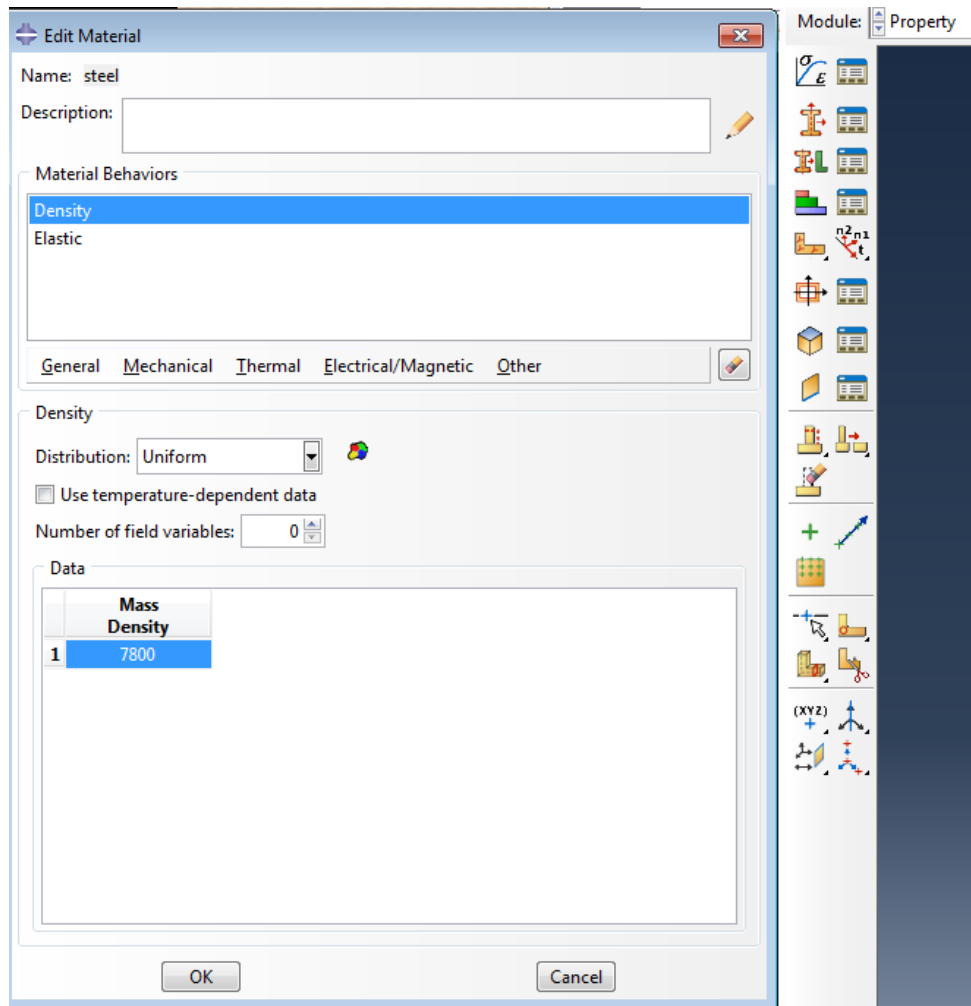
ModuleProperty

In the **Toolbox area**, click the **Create Material** icon. The **Create Material** dialog box appears. We will use the following steel:

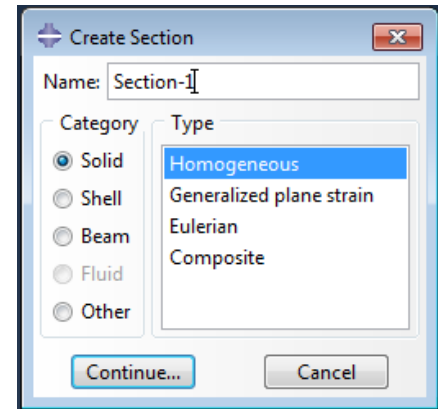
ASTM-A36 steel

$E = 2 \times 10^{11} \text{ Pa}$ *Young's modulus*

$\nu = 0.26$ *Poisson's ratio*

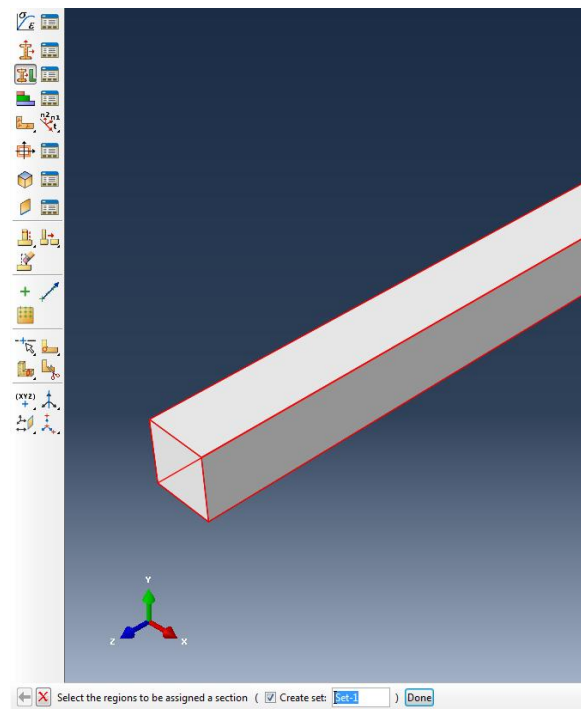


1. In the **Toolbox area**, click the **Create Section** icon. The **Create Section** dialog box appears. In the **Create Section** dialog box:
 - a. Name the section **BeamSection**.
 - b. In the **Category** list, select **Solid**.
 - c. In the **Type** list, select **Homogeneous**.
 - d. Click **Continue**.



2. The **Edit Section** dialog box appears. In the **Edit Section** dialog box:
 - a. Accept the default selection of **Steel** for the **Material** associated with the section. (If you had defined other materials, you could utilize the arrow beside the **Material** text box to select a material of your choice.)

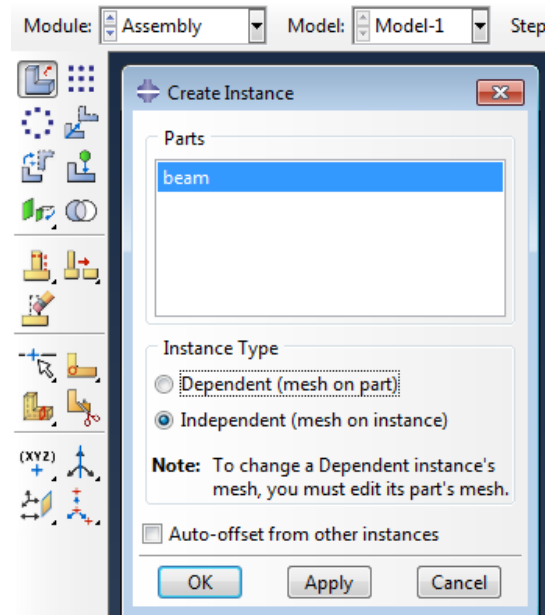
3. In the **Toolbox area**, click the **AssignSection** icon. The **AssignSection** dialog box appears. Select the entire part as the region to which the section will be applied. ABAQUS/CAE highlights the entire frame. Accept the default selection of **FrameSection**, and click **OK**.



Module Assembly

ABAQUS/CAE switches to the **Assembly** module.

1. Click the **Create Instance** icon. The **Create Instance** dialog box appears. In the dialog box:
 - a. Select **Frame** and click **OK**.

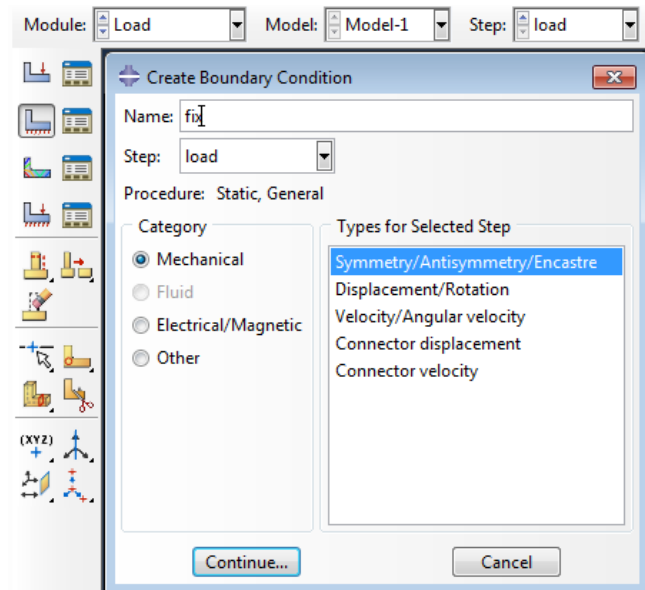


ModuleLoad

ABAQUS/CAE switches to the **Load** module.

1. Click the **Create Boundary Condition** icon. The **Create Boundary Condition** dialog box appears. In the **Create Boundary Condition** dialog box:

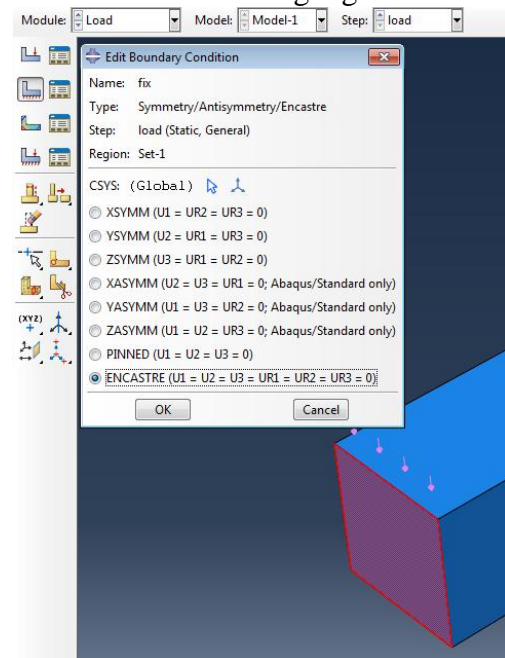
- a. Name the boundary condition **Fixed**.
- b. From the list of steps, select **Initial** as the step in which the boundary condition will be activated. (All mechanical boundary conditions specified in the Initial step must have zero magnitudes, a condition automatically enforced by ABAQUS/CAE.)
- c. In the **Category** list, accept **Mechanical** as the default category selection.
- d. In the **Types for Selected Step** list, select **Symmetry/Antisymmetry/Encastre**, and click **Continue**.



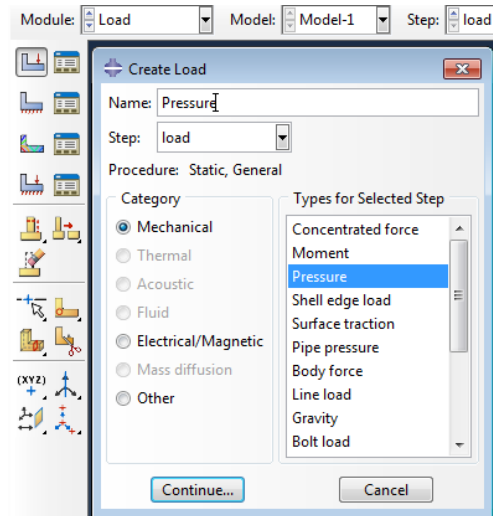
2. In the **viewport**, select one end surface of the beam as the region to which the boundary condition will be applied.
3. Click **Done** in the prompt area to indicate that you have finished selecting regions.

4. The **Edit Boundary Condition** dialog box appears. When you define a boundary condition in the initial step, all available degrees of freedom are unconstrained by default. In the dialog box:

- a. Toggle on **ENCASTRE**.
- b. Click **OK** to create the boundary condition and to close the dialog box.

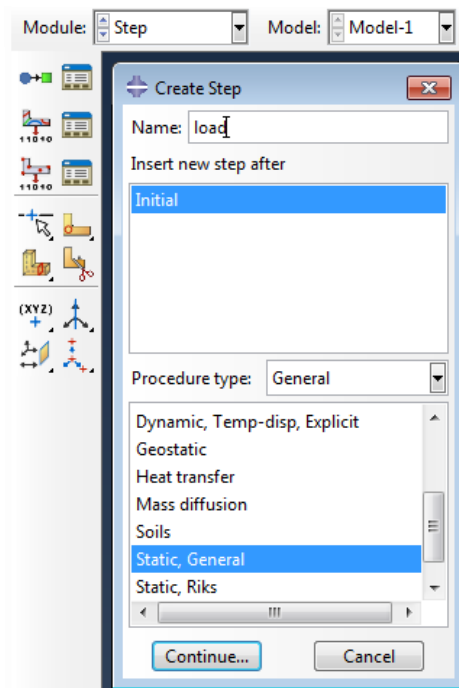


5. Click the **Create Load** icon. In the **Create Load** dialog box:
 - a. Name the load **Pressure**.
 - b. From the list of steps, select **Apply load** as the step in which the load will be applied.
6. In the **Category** list, accept **Mechanical** as the default category selection.
 - a. In the **Types for Selected Step** list, accept the default selection of **Pressure**.
 - b. Click **Continue**.



7. In the viewport, select the top surface of the beam as the region where the load will be applied, click **Done** in the prompt area to indicate that you have finished selecting regions.
8. The **Edit Load** dialog box appears. In the dialog box:
 - a. Enter a magnitude of **200.0**.
 - b. Click **OK** to create the load and to close the dialog box.

ABAQUS/CAE displays a downward-pointing arrow to indicate that the load is applied in the negative 2-direction.



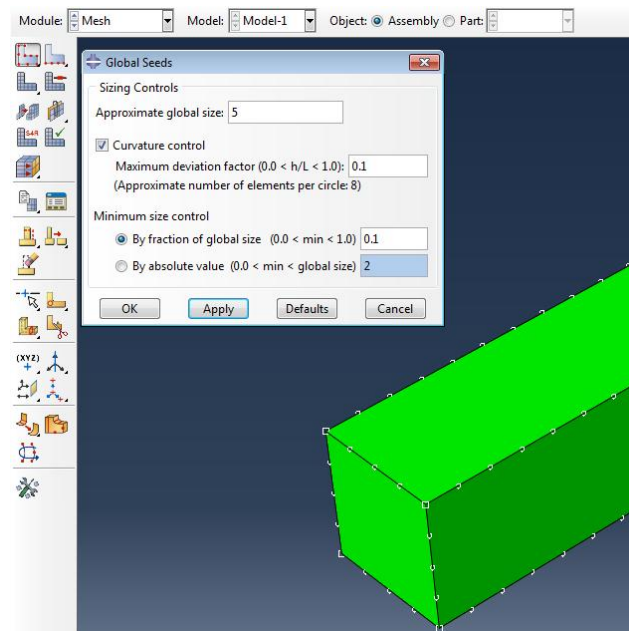
Module Mesh

ABAQUS/CAE switches to the **Mesh** module.

1. In the **menu bar**, select **Mesh**→**Element Type**.
2. In the **viewport**, drag the mouse to create a box that selects the entire beam as the region to be assigned an element type.
3. In the prompt area, click **Done** when you are finished.
4. The **Element Type** dialog box appears. Use these settings:
 - a. **Standard** as the **Element Library** selection (the default).
 - b. **Linear** as the **Geometric Order** (the default).
 - c. **3D** as the **Family** of elements.
 - d. Click **OK** to assign the element type and to close the dialog box.

To seed and mesh the model:

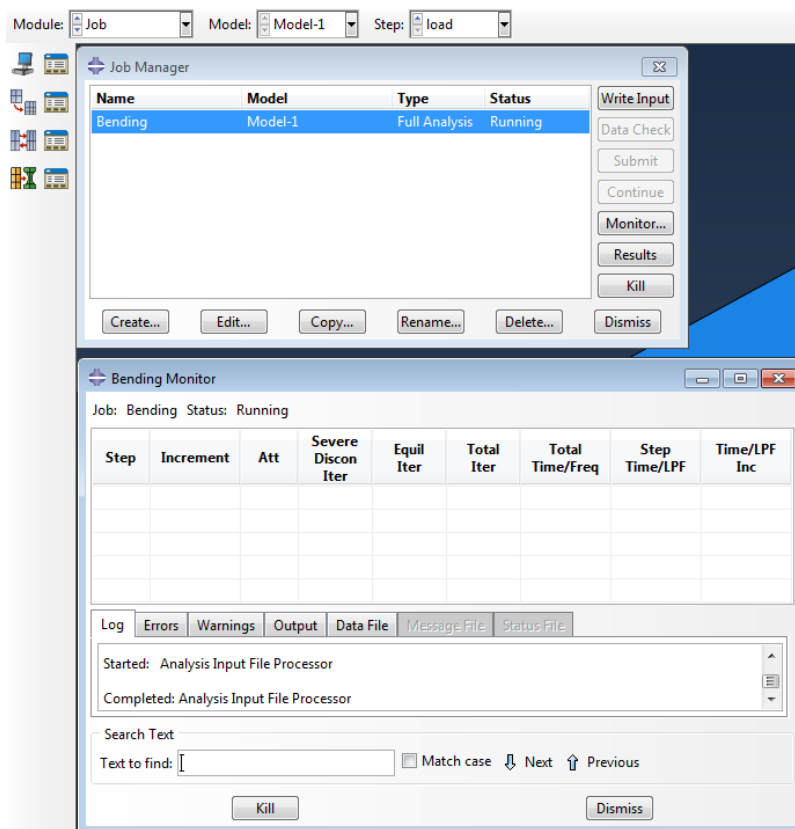
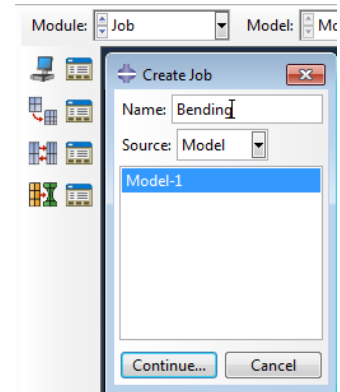
1. In the **menu bar**, select **Seed**→**Part** to seed the part instance. The **Global Seeds** dialog box appears.
 The dialog box displays the default element size that ABAQUS/CAE will use to seed the part instance. This default element size is based on the size of the part instance. A relatively large seed value will be used so that only one element will be created per region.
 - a. Specify an approximate global element size of **5.0**. Leave everything else as-is.
 - b. Click **OK** to create the seeds and to close the dialog box.
2. In the **menu bar**, select **Mesh**→**Part** to mesh the part instance.
3. From the buttons in the prompt area, click **Yes** to confirm that you want to mesh the part instance.



Module Job

ABAQUS/CAE switches to the Job module.

1. The **Create Job** dialog box appears with a list of the models in the model database. When you are finished defining your job, the **Jobs** container will display a list of your jobs.
 - a. Name the job **Beam**, and click **Continue**.
2. The **Edit Job** dialog box appears.
 - a. In the **Description** field, type Beam bending under pressure load.
3. Click **OK** to accept default job settings and to close the dialog box.



Module Visualization

You can also enter the Visualization module by selecting **Visualization** from the **Module** list located in the context bar.

