Rerunning the analysis using Abaqus/Explicit with postprocessing questions

Module*Property*

Modifying the material definition

Since Abaqus/Explicit performs a dynamic analysis, a complete material definition requires that you specify the material density. For this problem assume the density is equal to 7800 kg/m^3.

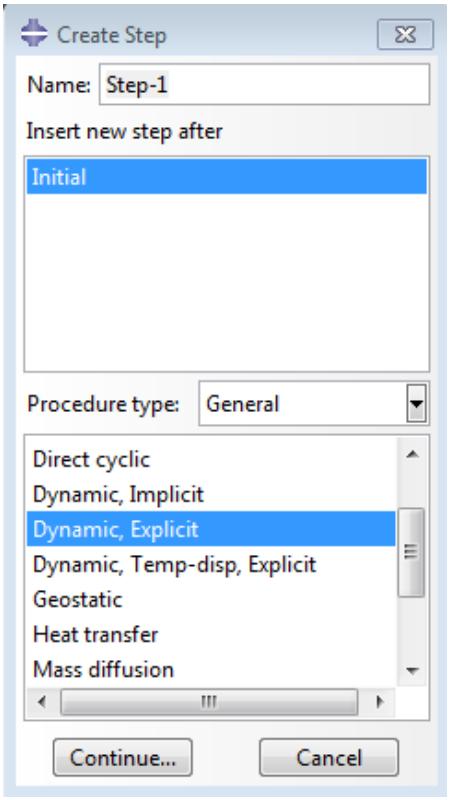
To add density to the material definition:

1. In the Model Tree, expand the **Materials** container and double-click **Steel**.
2. In the material editor, select **GeneralDensity** and type a value of `7800` for the density.
3. Click **OK** to close the **Edit Material** dialog box.

Module*Step*

The step definition must reflect a dynamic, explicit analysis.

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

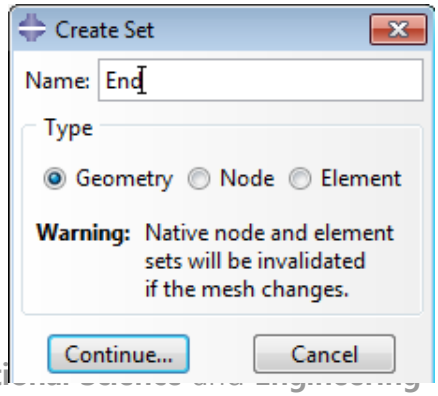


1. Click the **Create Step** icon. The **Create Step** dialog box appears. In the **Create Step** dialog box:
   a. Name the step `Load`.
   b. In the **Types for Selected Step** list, select `Dynamic`,Explict and click **Continue**.

1. In the **Basic** tabbed page of the **Edit Step** dialog box, enter the step description `suddenly applied` and set the time period of the step to `0.01` s.

Modifying the output requests

To create a set:



1. In the Model Tree, expand the **Assembly** container and double-click the **Sets** item.

   The **Create Set** dialog box appears.

2. Name the set `End`, accept the default selection of **Geometry**, and click **Continue**.

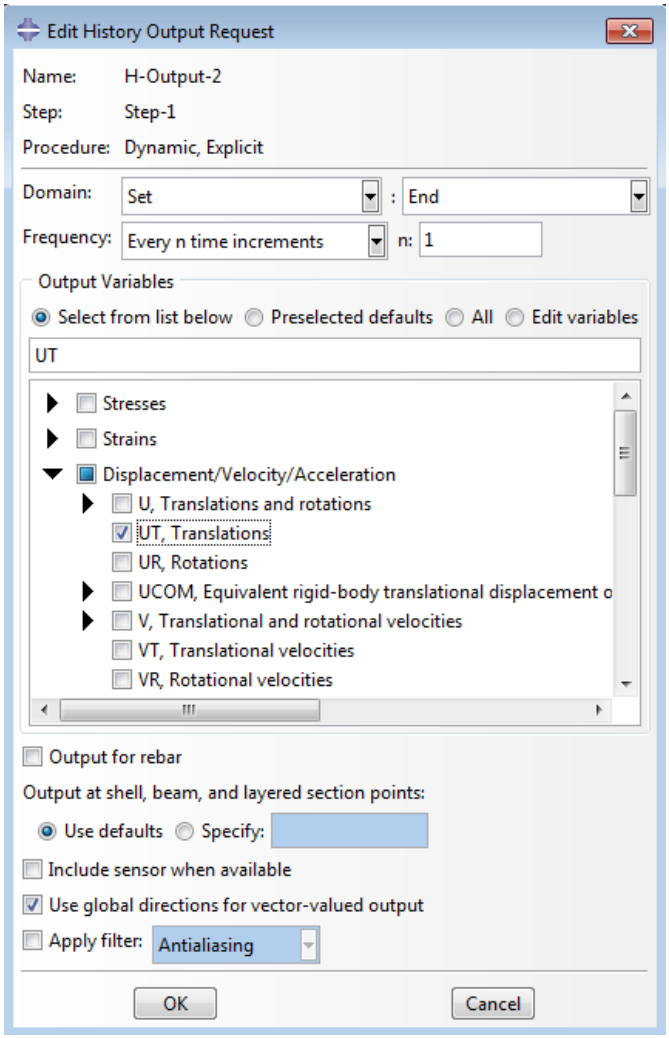
3. In the viewport, select the point at the end of the bottom edge of the beam. In the prompt area, click **Done** when you are finished.

To add displacements to the history output request:

1. Click on the **History Output Requests** icon, or in the **History Output Requests Manager** dialog box, click **Create**.

   The history output editor appears.

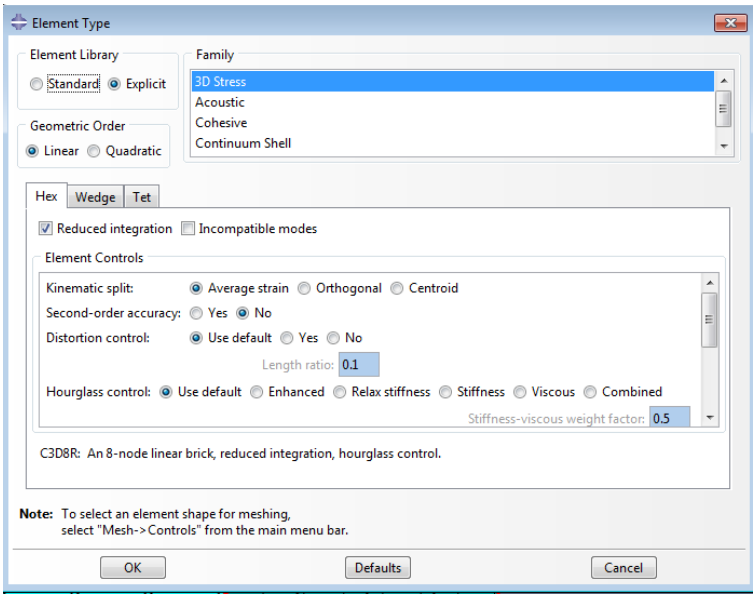
2. Under the **Domain** field, select **Set**. Abaqus automatically provides a list of all the sets created for a given model. In this case you have created only one set, `End`.
3. Under the **Frequency** field, select **Every n time increments** and set the value of **n** to `1` to write the output at every increment.
4. In the **Output Variables** region, click the arrow to the left of the **Displacement/Velocity/Acceleration** category to reveal history output options for translations and rotations.
5. Toggle on `UT, Translations` to have the displacements for the selected set be written as history output to the output database file.
6. Click **OK** to save your changes and to close the dialog box. Dismiss the **History Output Requests Manager**.



Module*Mesh*

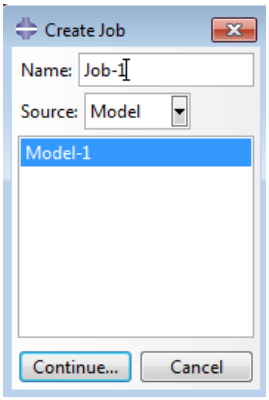
Changing the element library and submitting the job for analysis

1. From the main menu bar, select **MeshElement Type**, select the entire beam in the viewport, and change the **Element Library** selection to **Explicit**.
2. Click **OK** to accept the new element type.



Module*Job*

To create and run the new job



Postprocessing the dynamic analysis results

1, Create a time-history animation of the deformed model shape

2, Create an X–Y plot of the vertical displacement for a node