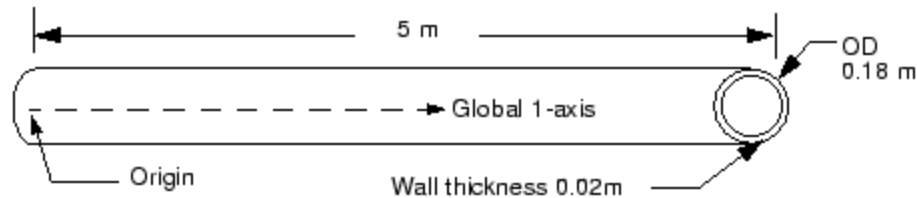


## 1 Vibration of a piping system

---

In this example you will study the vibrational frequencies of a 5m section of a piping system.



It is clamped firmly at one end and can **move only axially** at the other end. You will use three-dimensional beam elements to model the pipe section.

### Problem Description

The lowest vibrational mode of the unloaded structure is 40.1 Hz, but this value does not consider how the loading applied to the piping structure may affect its response. To ensure that the section does not resonate, you have been asked to determine the magnitude of the in-service load that is required so that its lowest vibrational mode is higher than 50 Hz. You are told that the section of pipe will be subjected to axial tension when in service. Start by considering a load magnitude of 4 MN.

## Preprocessing—creating the model with Abaqus/CAE


### Part geometry

Create a three-dimensional, deformable, planar wire part. (Remember to use an approximate part size that is slightly larger than the largest dimension in your model.) Name the part **Pipe**, and use the **Create Lines: Connected** tool to sketch a horizontal line of length 5.0 m.



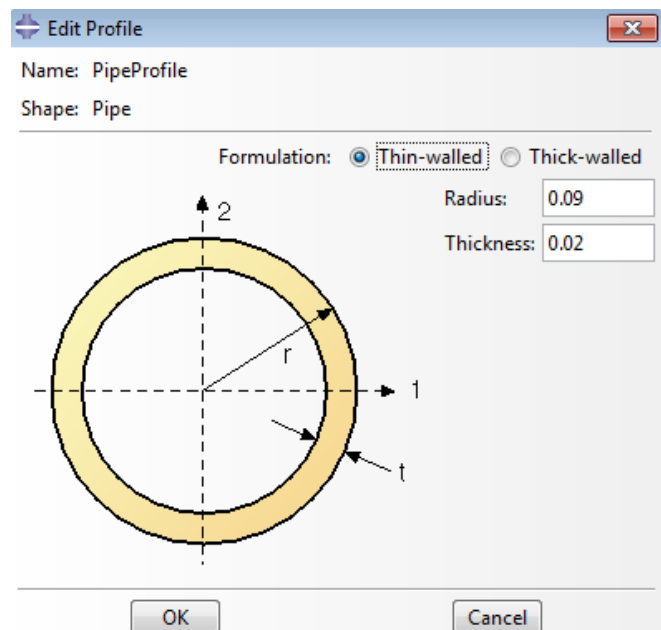
### Material and section properties

The pipe is made of steel with a Young's modulus  $E = 2 \times 10^{11}$  Pa and a Poisson's ratio  $\nu = 0.3$ . Create a linear elastic material named **Steel** with these properties. The density of the steel material is  $\rho = 7800$  kg/m<sup>3</sup>.

Next, create a **Pipe** profile from the main menu bar **Profile** or from the toolbar box .

Name the profile **PipeProfile**, and specify an outer radius of 0.09 m and a wall thickness of 0.02 m for the pipe.

Create a **Beam** section named **PipeSection**. In the **Edit Beam Section** dialog box, specify that section integration will be performed during the analysis and assign material **Steel** and profile **PipeProfile** to the section definition. Finally, assign section **PipeSection** to the pipe.



### Assembly and sets

Create a **dependent** instance of the part named **Pipe**.

Create geometry sets that contain the points at the left and right ends of the pipe and name them **Left** and **Right**, respectively.

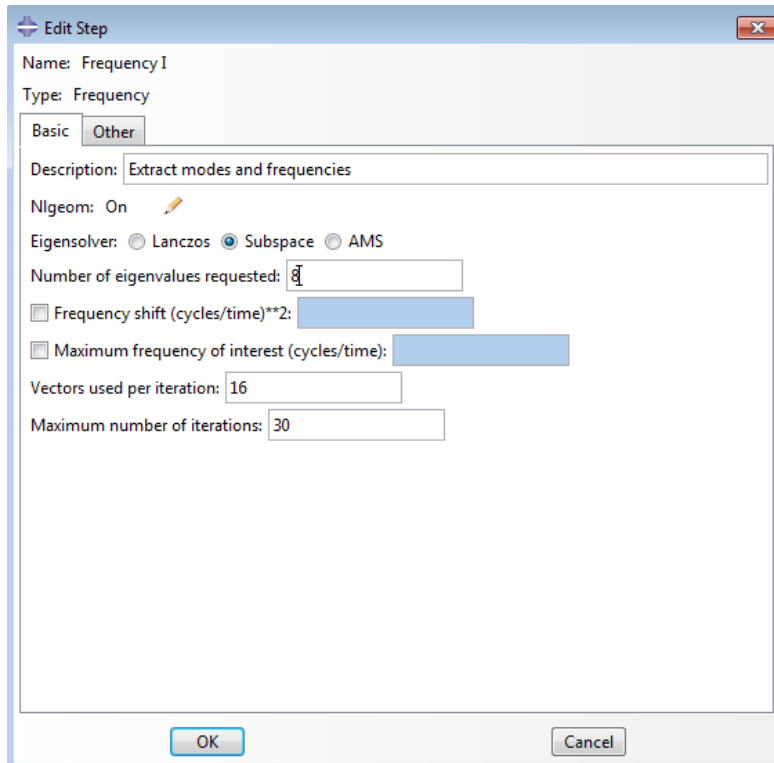
## Module Steps

*Step 1.* General step: Apply a 4 MN tensile force.

*Step 2.* Linear perturbation step: Calculate modes and frequencies.

Create a general static step named Pull I with the following step description: Apply axial tensile load of 4.0 MN.

*Timestep 1.* Include the effects of **geometric nonlinearity** and specify an **initial increment size** that is 1/10 the total step time.



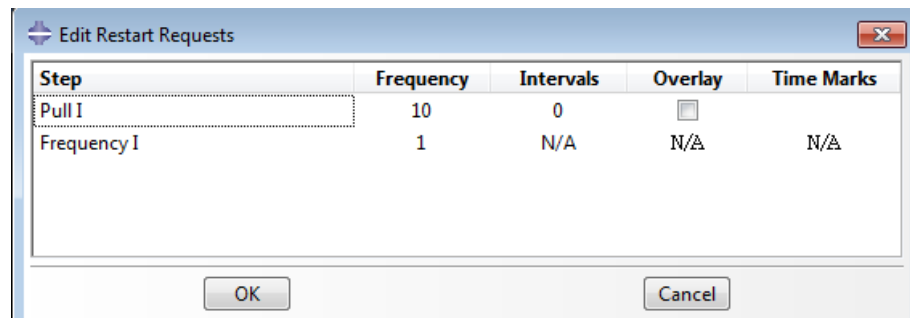
*Second analysis step: linear perturbation* frequency extraction procedure. Name the step Frequency I, and give it the following description: Extract modes and frequencies. Extract the first 8 eigenmodes for the model.

### Output requests

The default output database output requests created by Abaqus/CAE for each step will suffice; you do not need to create any additional output database output requests.

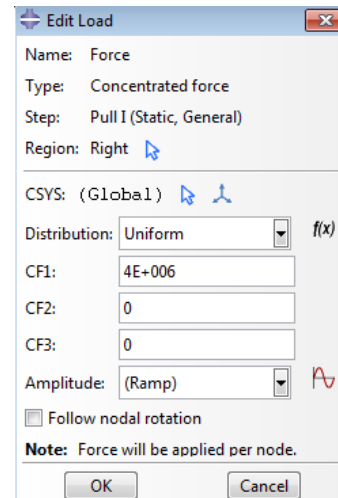
To request output to the restart file, select **Output** → **Restart Requests** from the main menu bar of the Step module. For the step labeled Pull I, write data to the restart file every 10 increments; for the step

labeled Frequency I, write data to the restart file every increment.

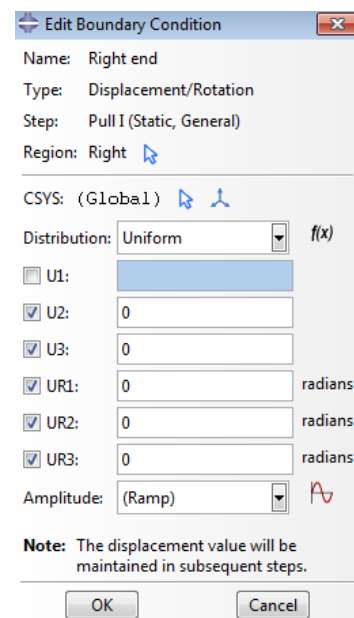


## Loads and boundary conditions

In the first step create a load named Force that applies a 4E6 N tensile force to the right end of the pipe section such that it deforms in the positive axial (global 1) direction.



The pipe section is clamped at its left end. It is also clamped at the other end; however, the axial force must be applied at this end, so only degrees of freedom 2 through 6 (U2, U3, UR1, UR2, and UR3) are constrained.



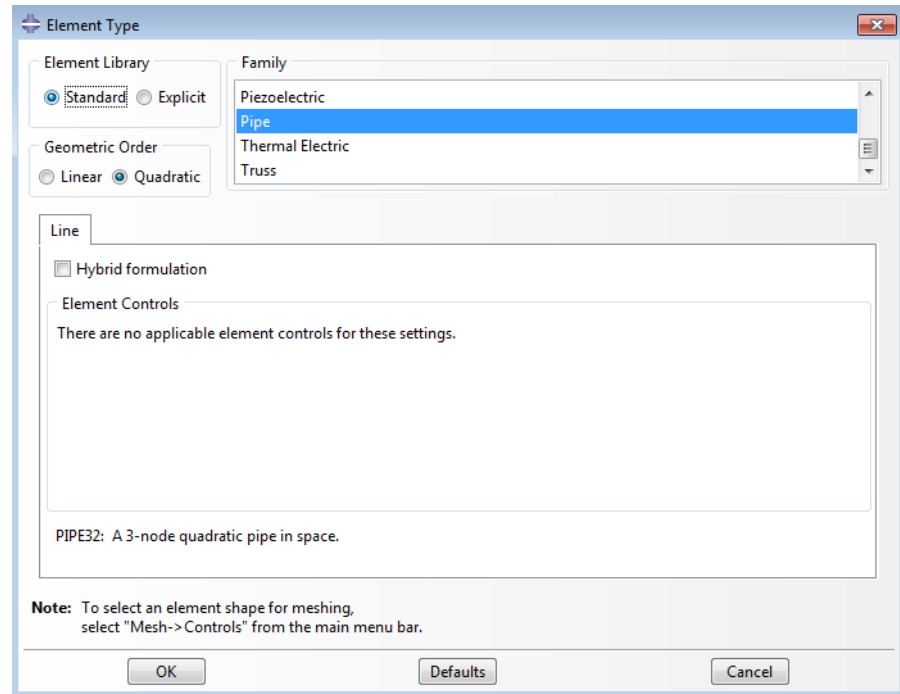
## Mesh and job definition

Seed and mesh the pipe section with 30 uniformly spaced second-order, pipe elements (PIPE32).

Before continuing, rename the model to Original.

Create a job named Pipe with the description “Analysis of a 5 meter long pipe under tensile load”.

Save your model in a model database file, and submit the job for analysis.



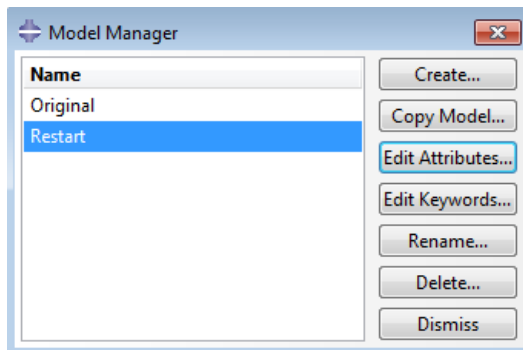
## Postprocessing

Enter the **Visualization** module, and open the output database file, Pipe.odb, created by this job.

**To plot the first mode shape:**

1. From the main menu bar, select **Result**→**Step/Frame**.
2. In the **Step/Frame** dialog box, select step Frequency 1 and frame Mode 1.
3. Click **OK**.

## 2 Restart analysis of vibration of a piping system

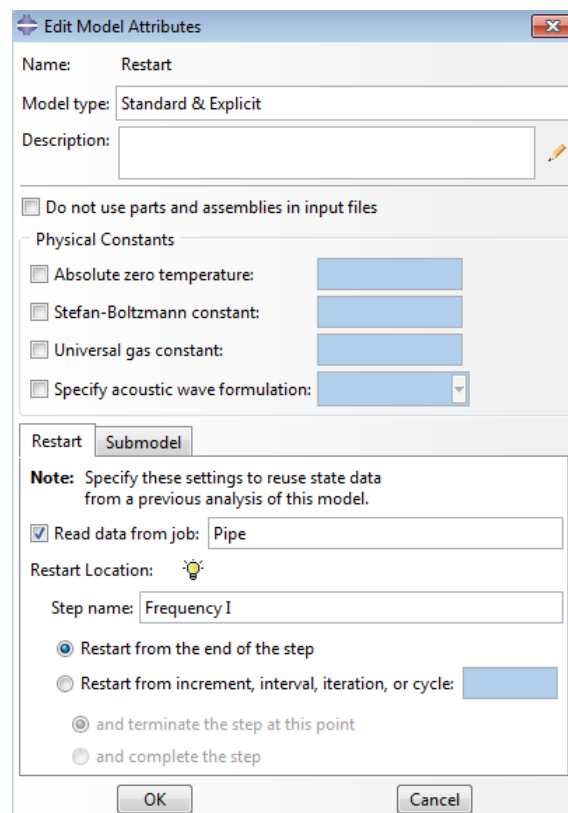


Copy the model named Original to a model named Restart.

### Model attributes

To perform a restart analysis, the model's attributes must be changed to indicate that the model should reuse data from a previous analysis.

1. In the **Model Tree**, double-click the **Restart** model underneath the **Models** container.
2. In the **Edit Model Attributes** dialog box that appears, specify that restart data will be read from the job Pipe and that the restart location will be the end of step Frequency I.

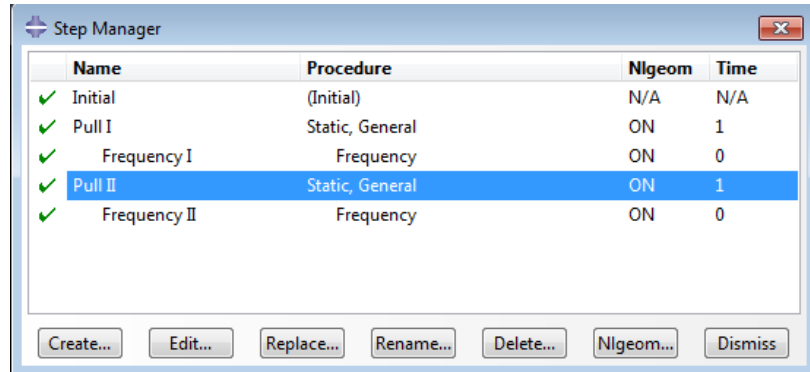


## Step definitions

You will now create two new analysis steps.

1. The first new step is a general static step.

- a. Name the step Pull II, and insert it immediately after the step Frequency I.
- b. Give the step the description “Apply axial tensile load of 8.0 MN”.
- c. Set the time period for the step to 1.0 and the initial time increment to 0.1.



2. The second new step is a frequency extraction step.

- a. Name the step Frequency II.
- b. Insert it immediately after the step Pull II.
- c. Give the step the description “Extract modes and frequencies”.

## Output requests

3. For the step Pull II, write data to the restart file every 10 increments.
4. Accept all other default output data requests.

## Load definition

5. Modify the load definition so that the tensile load that is applied to the pipe is doubled in the second general static step (Pull II). To do this,
  - a. Expand the **Force** item underneath the **Loads** container in the Model Tree.
  - b. In the list that appears, expand the **States** item.
  - c. Double-click the step named **Pull II**.
  - d. Change the value of the applied force to 8.0E+06 in this step.

## Job definition

6. Create a job named PipeRestart with the description “Restart analysis of a 5 meter long pipe under tensile load”.
7. Set the job type to **Restart** if it is not already. (If the job type is not set to **Restart**, Abaqus/CAE ignores the model's restart attributes.)
8. Save your model in a model database file, and submit the job for analysis.
  - a. Monitor solution progress; correct any modeling errors and investigate the source of any warning messages, taking corrective action as necessary.

## Postprocessing the restart analysis results

---

### Plotting the eigenmodes of the pipe

### Plotting X–Y graphs from field data for selected steps

Use the field data stored in the output database files, `Pipe.odb` and `PipeRestart.odb`, to plot the history of the axial stress in the pipe for the whole simulation.

**To generate a history plot of the axial stress in the pipe for the restart analysis:**

1. In the Results Tree, double-click **XYData** to show the **Create XY Data** dialog box.
  - a. Select **ODB field output** from this dialog box, and click **Continue** to proceed.
2. In the **XY Data from ODB Field Output** dialog box:
  - a. In the **Variables** tabbed page of this dialog box:
    - i. Accept the default selection of Integration Point for variable position.
    - ii. Select S11 from the list of available stress components.
    - iii. Toggle **Select** for the section point.
    - iv. Click **Settings** to choose a section point.
3. In the **Field Report Section Point Settings** dialog box:
  - a. Select the category beam and choose any available section point for the pipe cross-section.
4. In the **Elements/Nodes** tabbed page of the **XY Data from ODB Field Output** dialog box:
  - a. Select `Element labels` as the selection **Method**.



There are 30 elements in the model, and they are numbered consecutively from 1 to 30. Enter any element number (*e.g.*, 18) in the **Element labels** field.

5. Click **Active Steps/Frames**, and select `Pull II` as the only step to extract data from.
6. At the bottom of the **XY Data from ODB Field Output** dialog box:
  - a. Click **Plot** to see the history of axial stress in the element.

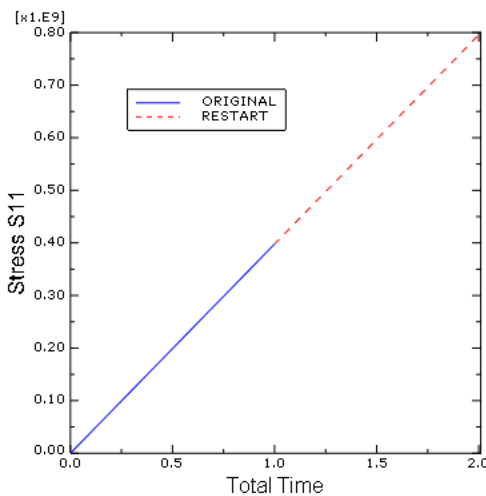
The resulting plot shows the axial stress history for each integration point of the element during the restart analysis. Since there is a job history prior to the restart, it is desirable to view the entire analysis.



To generate a history plot of the axial stress in the pipe for the entire analysis:

1. Save the current plot by clicking **Save** at the bottom of the **XY Data from ODB Field Output** dialog box. Two curves are saved (one for each integration point), and default names are given to the curves.
2. Rename one curve RESTART, and delete the other curve.
3. From the main menu bar, select **File**→**Open** or use the  tool in the **File** toolbar to open the file Pipe.odb.
4. Following the procedure outlined above, save the plot of the axial stress history for the same element and integration/section point used above. Name this plot ORIGINAL.
5. In the **Results Tree**, expand the **XYData** container.
6. Select both plots with **[Ctrl]+Click**. Click mouse button 3 to display a context menu. Select **Plot** from this menu to create a plot of axial stress history in the pipe for the entire simulation.
7. To change the style of the line, open the **Curve Options** dialog box.
  1. For the RESTART curve, select a dotted line style .
8. **Dismiss** the dialog box.
9. To change the axis titles, open the **Axis Options** dialog box. In the **Title** tab:
  1. Change the X-axis title to TOTAL TIME.
  2. Change the Y-axis title to STRESS S11.
10. **Dismiss** the dialog box.

History of axial stress in the pipe.



History of axial stress in the pipe during Step 3.

