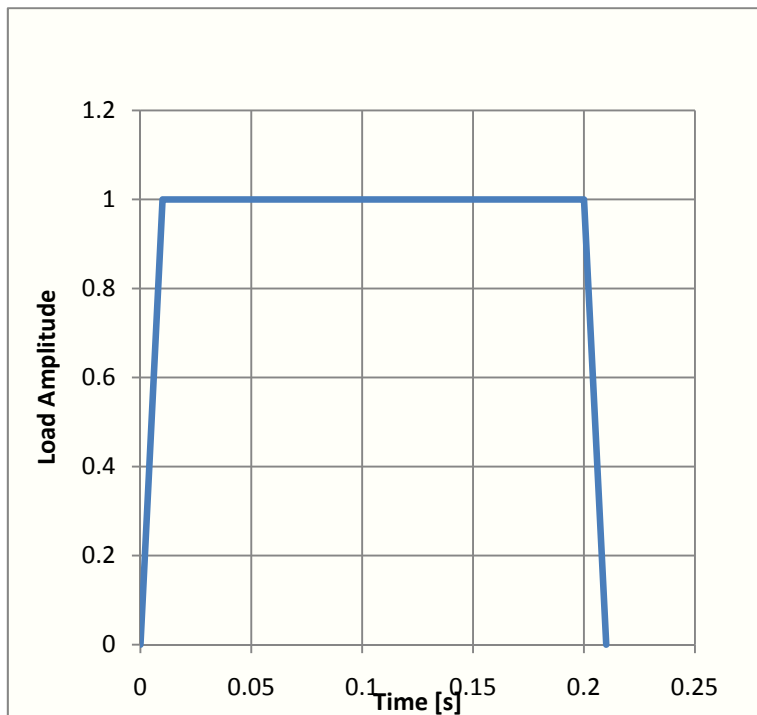
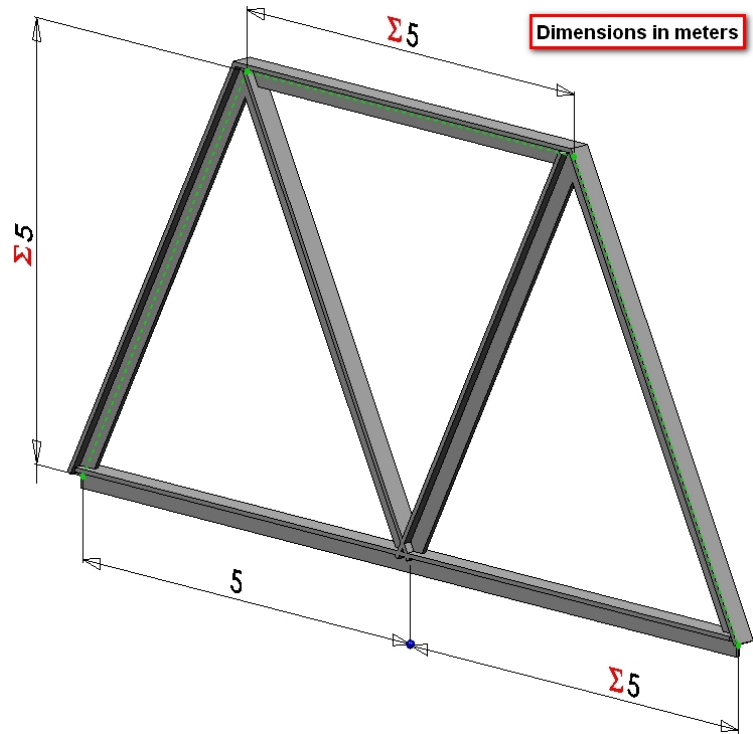


Abaqus/CAE Dynamic Response Tutorial

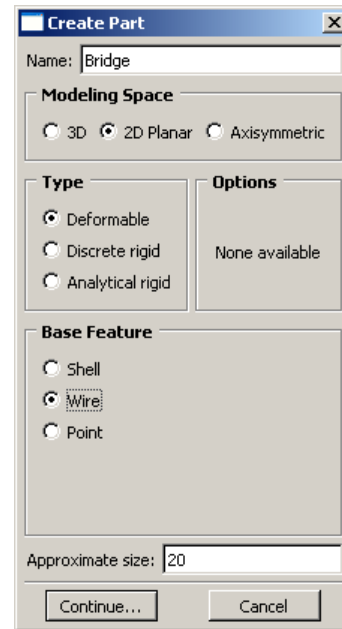
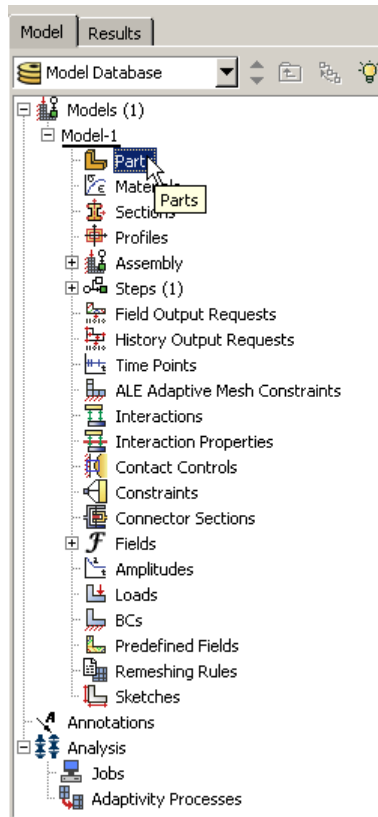
Problem Description

The two dimensional bridge structure, which consists of steel sections, is simply supported at its lower corners, and subjected to the following time varying load.

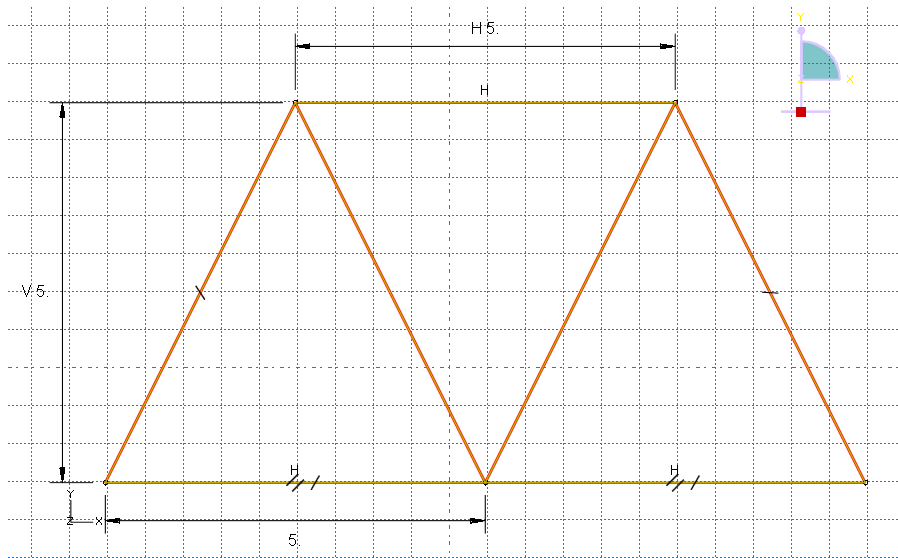


Analysis Steps

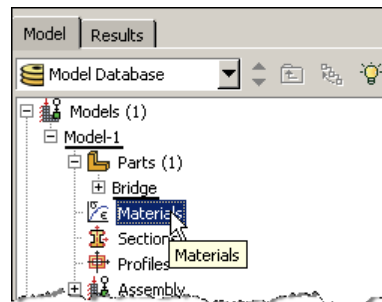
1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)



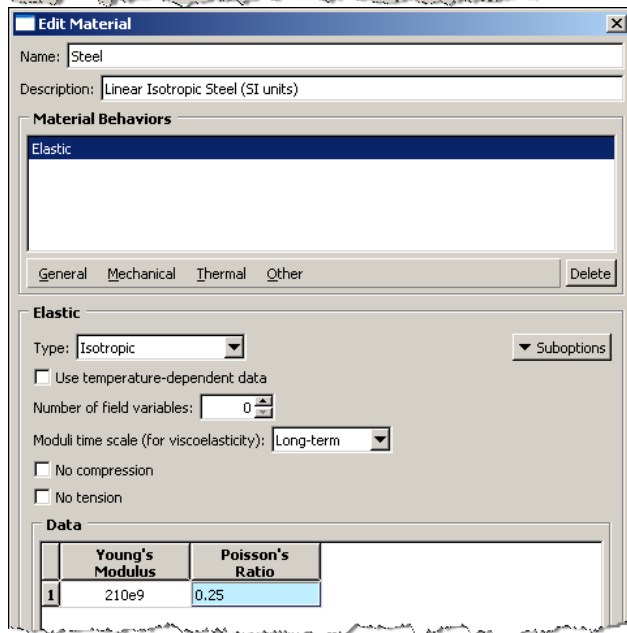
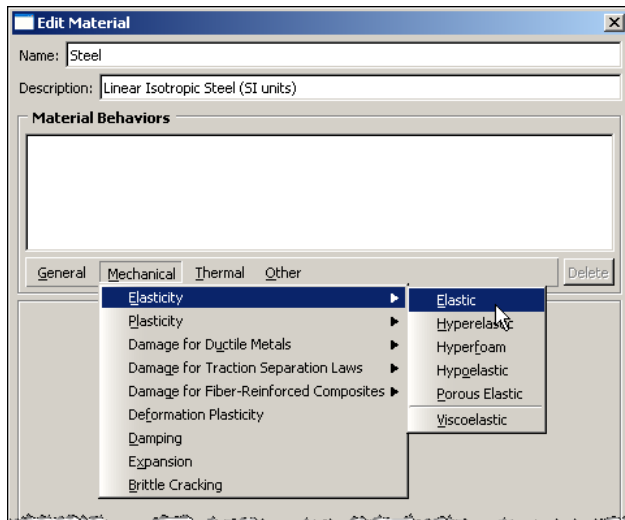
3. In the Create Part dialog box (shown above) name the part and
 - a. Select “2D Planar”
 - b. Select “Deformable”
 - c. Select “Wire”
 - d. Set approximate size = 20
 - e. Click “Continue...”
4. Create the geometry shown below (not discussed here)



5. Double click on the “Materials” node in the model tree

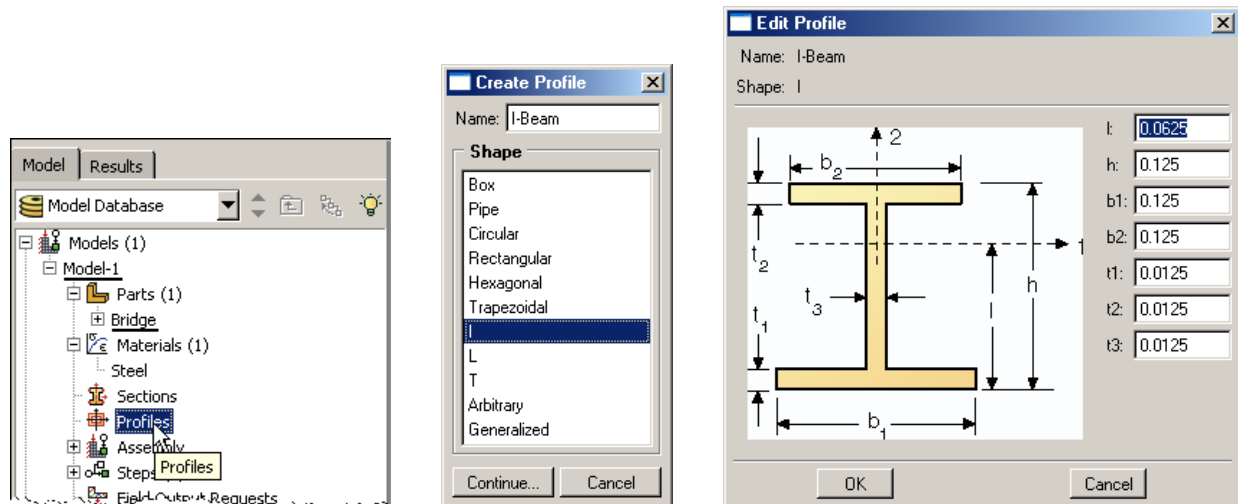


- Name the new material and give it a description
- Click on the “Mechanical” tab → Elasticity → Elastic
- Define Young’s Modulus and Poisson’s Ratio (use SI units)

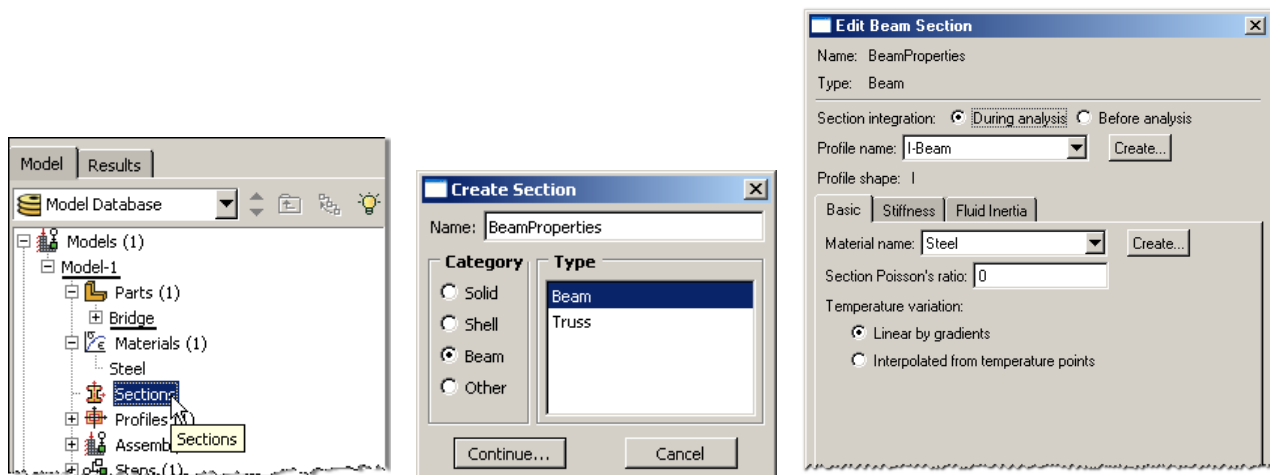


- d. Click on the "General" tab → Density
- e. Density = 7800
- f. Click "OK"

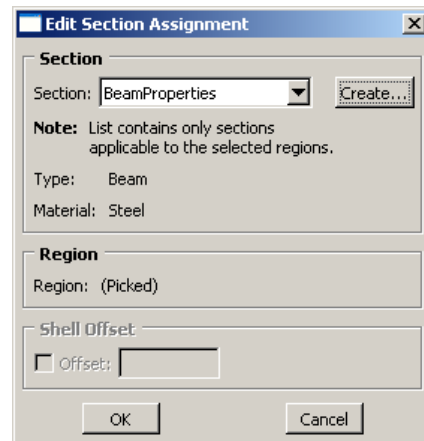
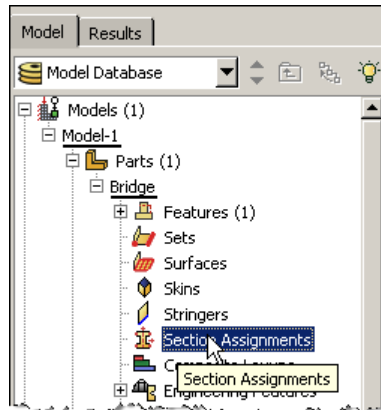
6. Double click on the “Profiles” node in the model tree
 - a. Name the profile and select “I” for the shape
 - b. Click “Continue...”
 - c. Enter the values for the profile shown below
 - d. Click “OK”



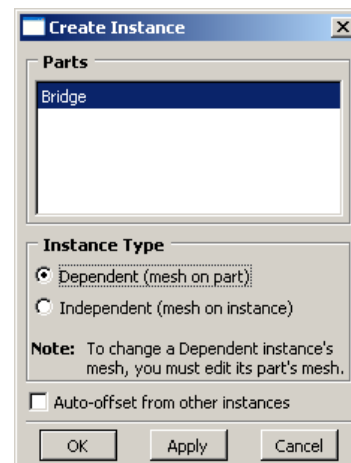
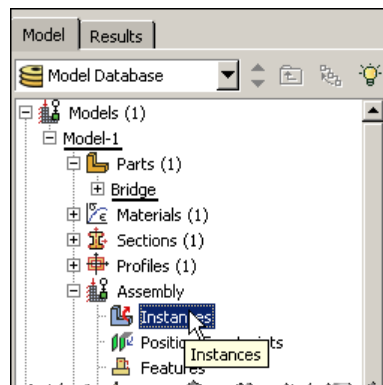
7. Double click on the “Sections” node in the model tree
 - a. Name the section “BeamProperties” and select “Beam” for both the category and the type
 - b. Click “Continue...”
 - c. Leave the section integration set to “During Analysis”
 - d. Select the profile created above (I-Beam)
 - e. Select the material created above (Steel)
 - f. Click “OK”



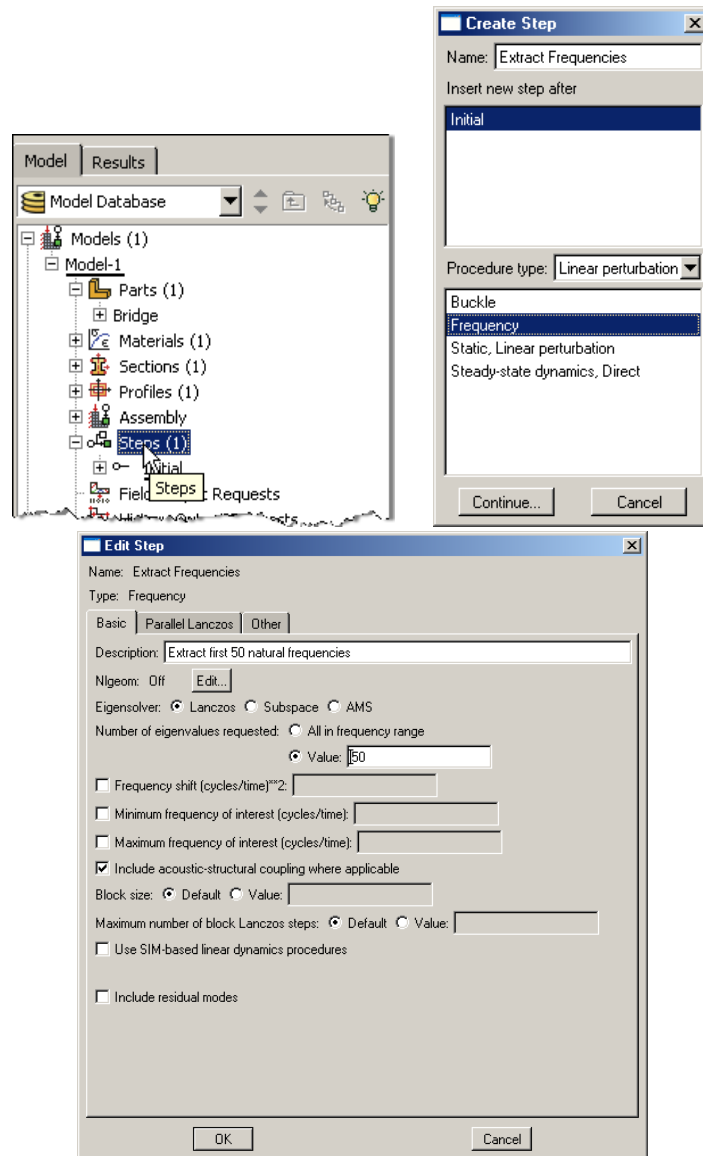
8. Expand the “Parts” node in the model tree, expand the node of the part just created, and double click on “Section Assignments”
- Select the entire geometry in the viewport
 - Select the section created above (BeamProperties)
 - Click “OK”



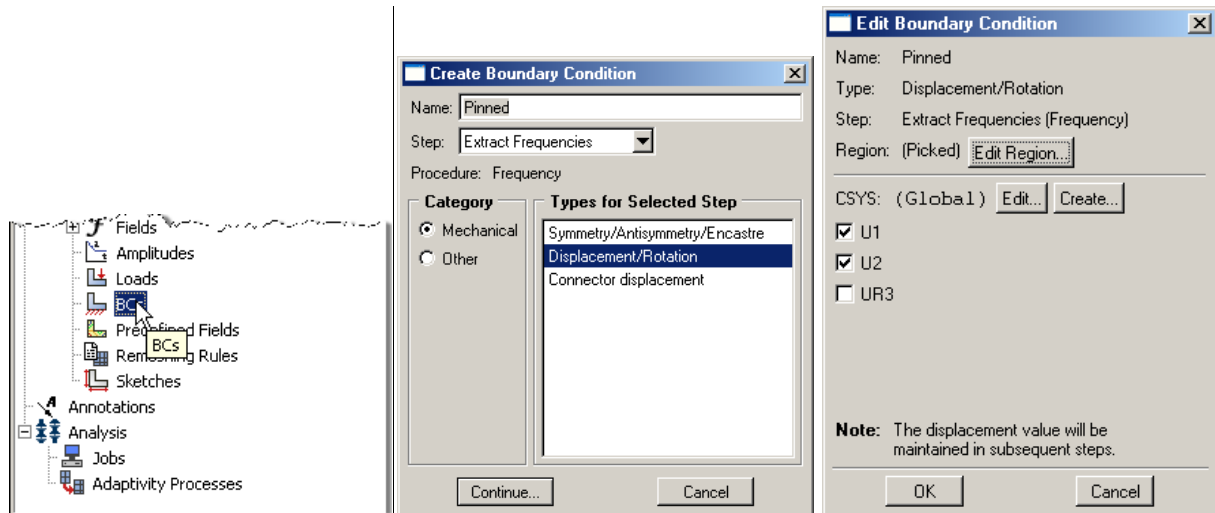
9. Expand the “Assembly” node in the model tree and then double click on “Instances”
- Select “Dependent” for the instance type
 - Click “OK”



10. Double click on the “Steps” node in the model tree
 - a. Name the step, set the procedure to “Linear perturbation”, and select “Frequency”
 - b. Click “Continue...”
 - c. Give the step a description
 - d. Number of eigenvalues requested = 50
 - e. Click “OK”

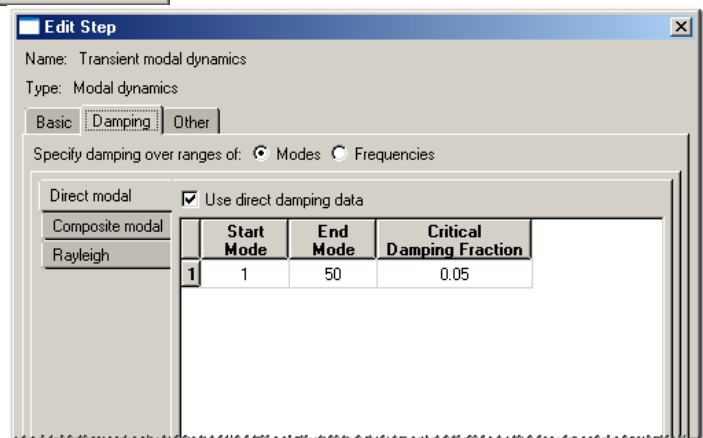
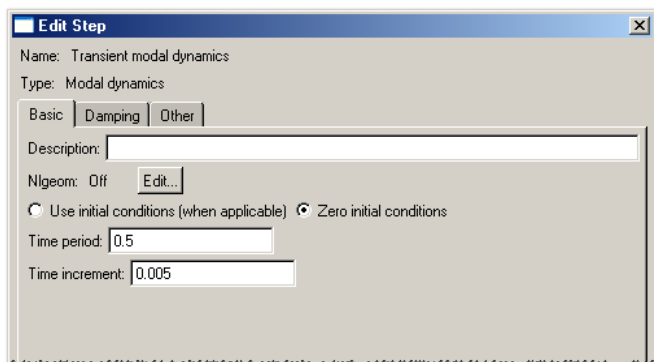
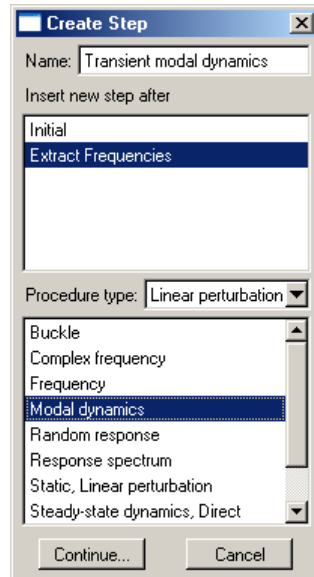


11. Double click on the “BCs” node in the model tree
- Name the boundary conditioned “Pinned” and select “Displacement/Rotation” for the type
 - Click “Continue...”
 - Select the lower-left and lower-right corners of the geometry and press “Done” in the prompt area
 - Check the U1 and U2 displacements and set them to 0
 - Click “OK”



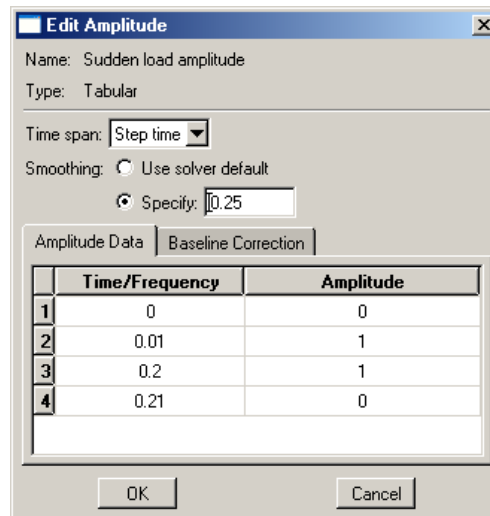
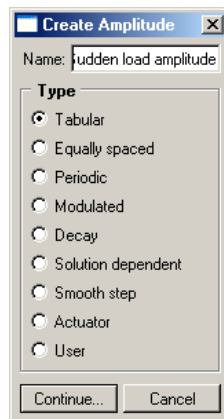
12. Double click on the “Steps” node in the model tree

- Name the step, set the procedure to “Linear perturbation”, and select “Modal dynamics”
- Click “Continue...”
- On the basic tab, set the time period to 0.5 and the time increment to 0.005 (total of 100 increments)
- On the damping tab, apply a critical damping fraction of 0.05 to all 50 modes

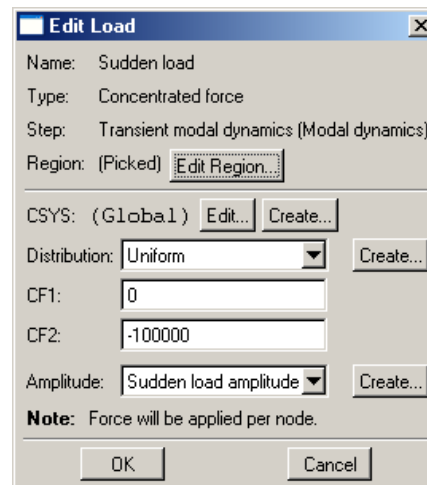
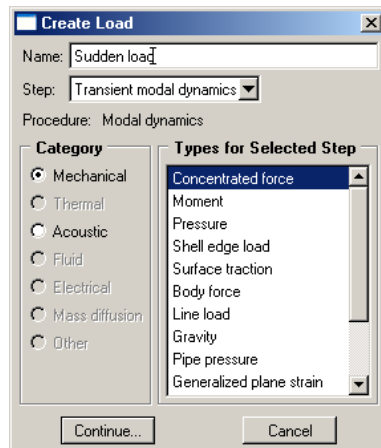


13. In the model tree double click on “Amplitudes”

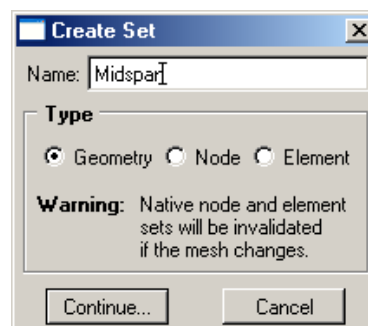
- Select “Tabular” and name the amplitude
- Specify a smoothing factor of 0.25
- Enter the time and amplitudes shown below



14. Double click on the “Loads” node in the model tree
 - a. Select “Concentrated force” and click Continue...
 - b. Select the point in the middle of the lower member
 - c. Set CF2 to -100000
 - d. Choose the amplitude created above and click “OK”

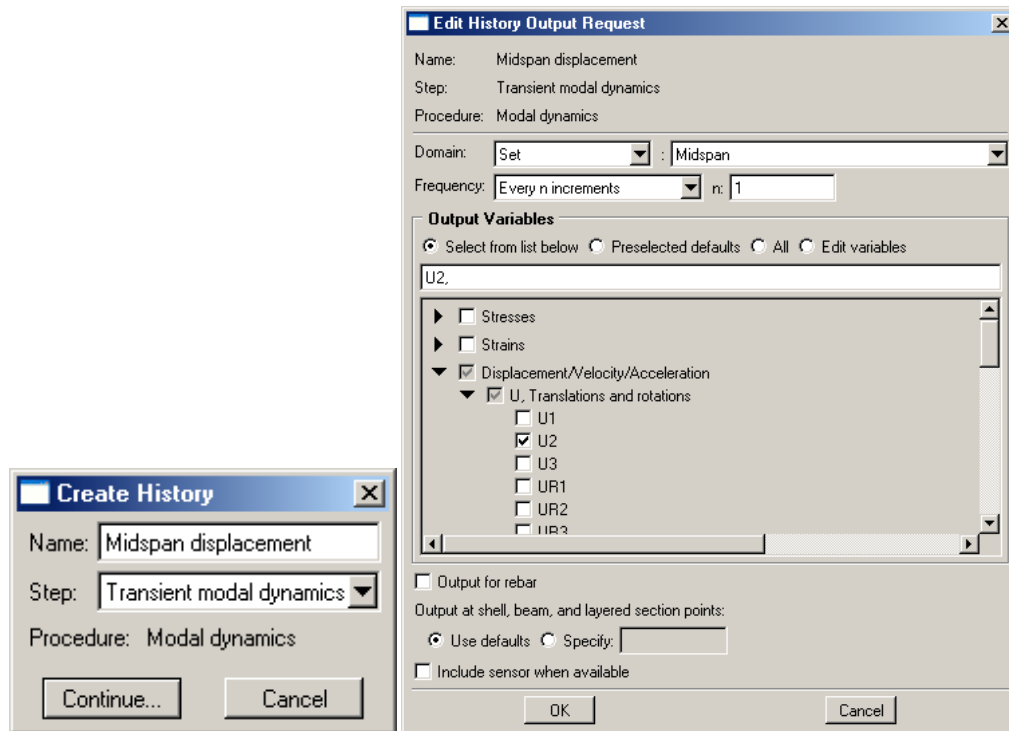


15. In the model tree under the “Assembly” node, double click on “Sets”
 - a. Name the set, select the “Geometry” option, and click Continue...
 - b. Select the point that the load was applied to

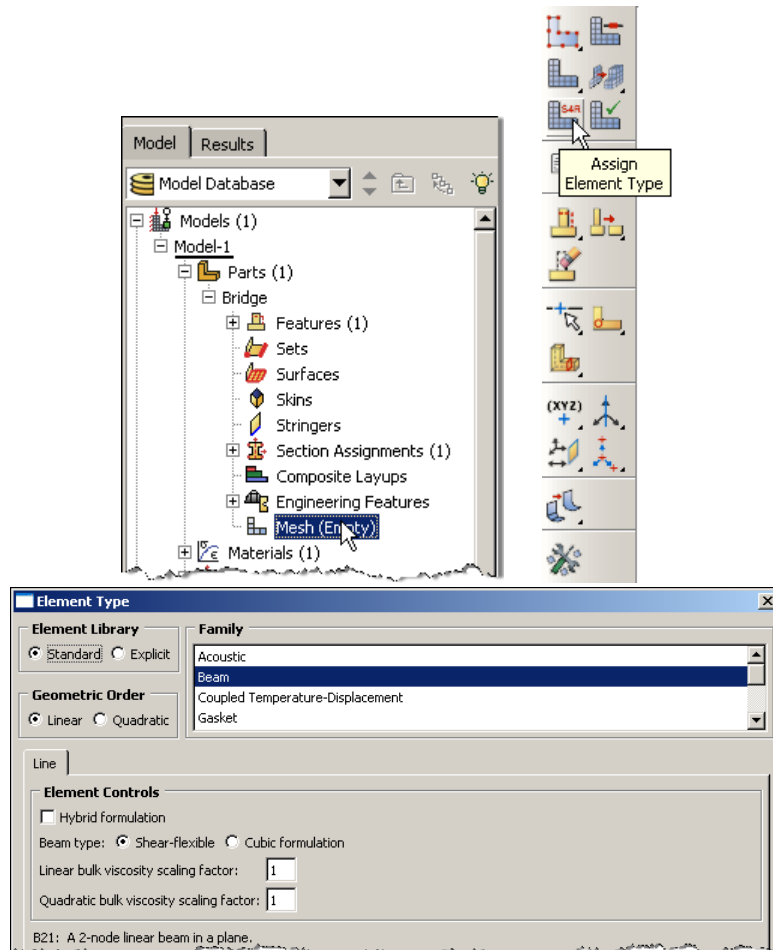


16. Double click on the “History output requests” node in the model tree
 - a. Name the history output, select the modal dynamics step, and click Continue...
 - b. Set the domain to “Sets” and select the set created above

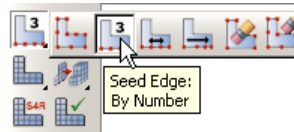
- c. For the frequency choose to output every 1 increments
- d. For the output select the U2 displacement
- e. Click OK



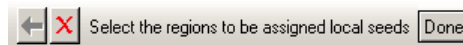
17. In the model tree double click on “Mesh” for the Bridge part, and in the toolbox area click on the “Assign Element Type” icon
 - a. Select “Standard” for element type
 - b. Select “Linear” for geometric order
 - c. Select “Beam” for family
 - d. Note that the name of the element (B21) and its description are given below the element controls
 - e. Click “OK”



18. In the toolbox area click on the “Seed Edge: By Number” icon (hold down icon to bring up the other options)



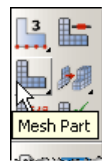
a. Select the entire geometry and click “Done” in the prompt area



b. Define the number of elements along the edges as 20

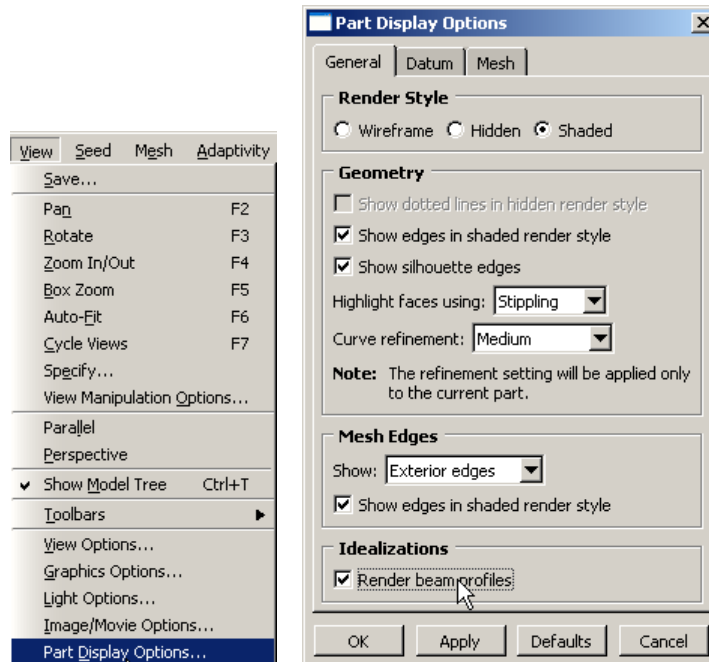
19. In the toolbox area click on the “Mesh Part” icon

a. Click “Yes” in the prompt area



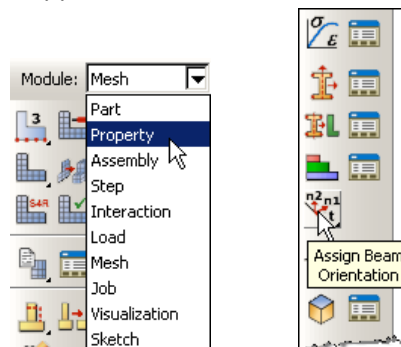
20. In the menu bar select View→Part Display Options

- Check the Render beam profiles option
- Click “OK”



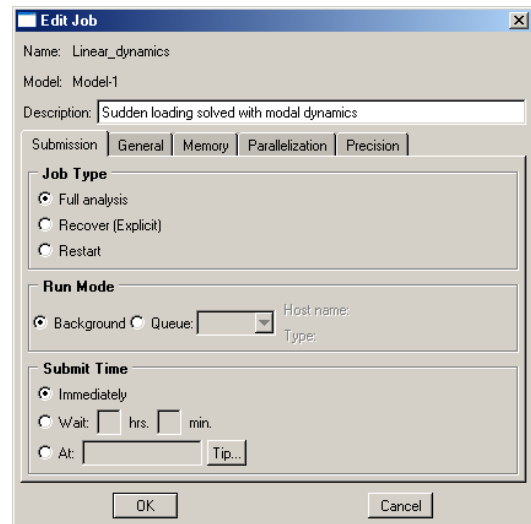
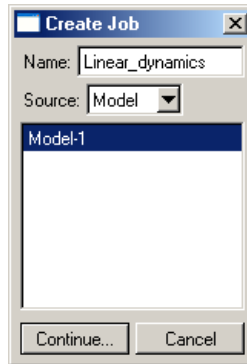
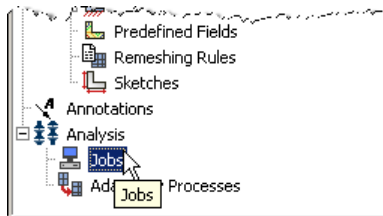
21. Change the Module to “Property”

- Click on the “Assign Beam Orientation” icon
- Select the entire geometry from the viewport
- Click “Done” in the prompt area
- Accept the default value of the approximate n1 direction

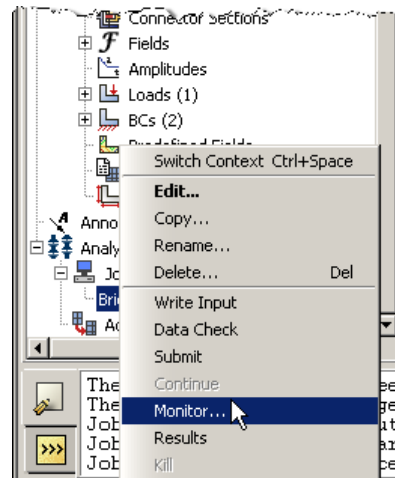
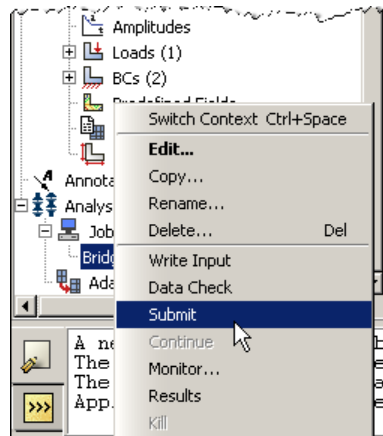


22. In the model tree double click on the “Job” node

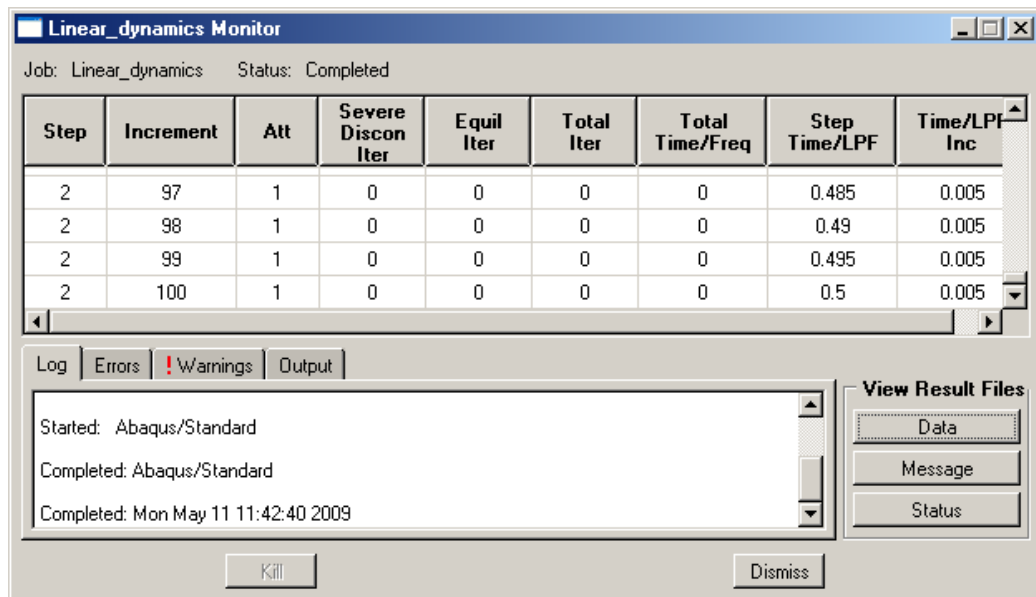
- Name the job
- Click “Continue...”
- Give the job a description
- Click “OK”



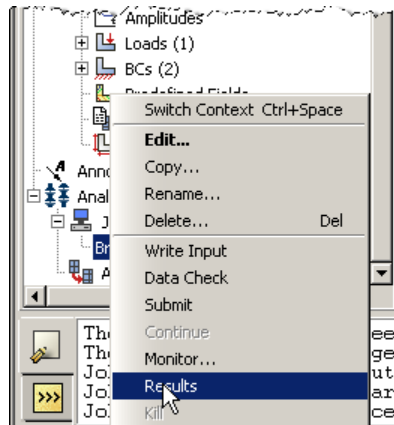
23. In the model tree right click on the job just created (Bridge) and select “Submit”
- While Abaqus is solving the problem right click on the job submitted (Bridge), and select “Monitor”



- b. In the Monitor window check that there are no errors or warnings
 - i. If there are errors, investigate the cause(s) before resolving
 - ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored



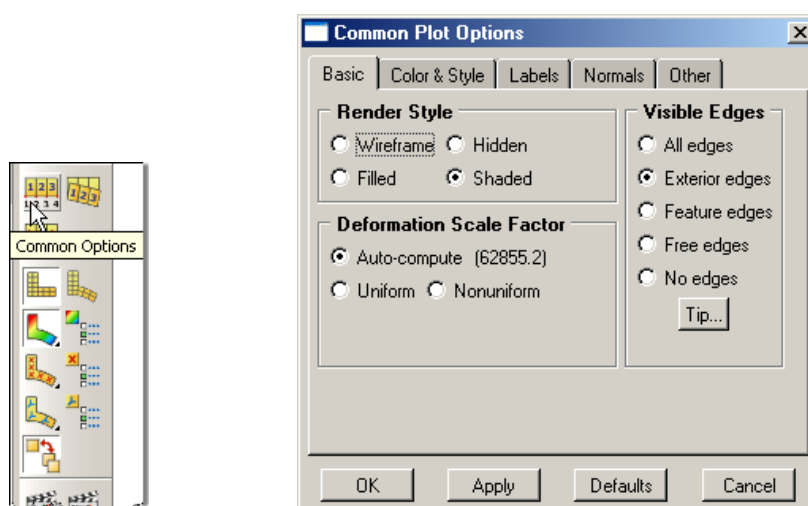
24. In the model tree right click on the submitted and successfully completed job , and select “Results”



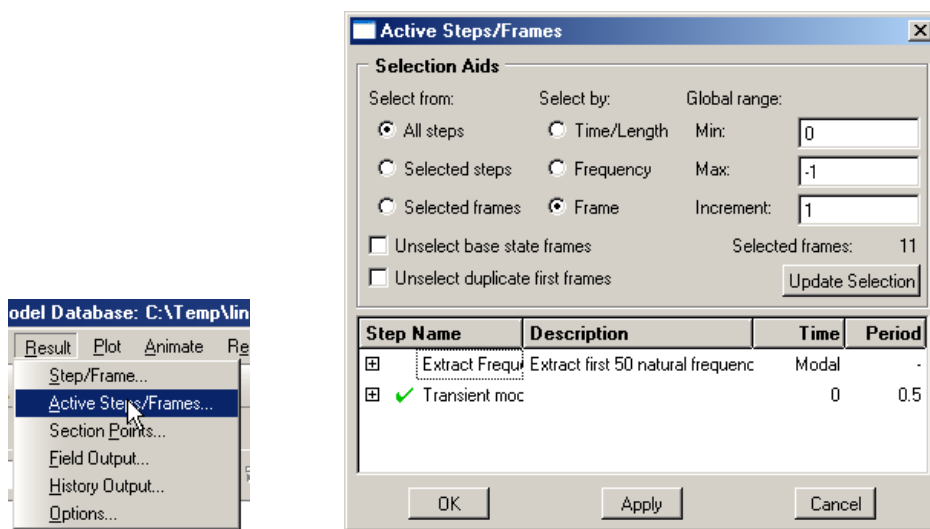
25. Display the deformed contour overlaid with the undeformed geometry
 - a. In the toolbox area click on the following icons
 - i. “Plot Contours on Deformed Shape”
 - ii. “Allow Multiple Plot States”
 - iii. “Plot Undeformed Shape”



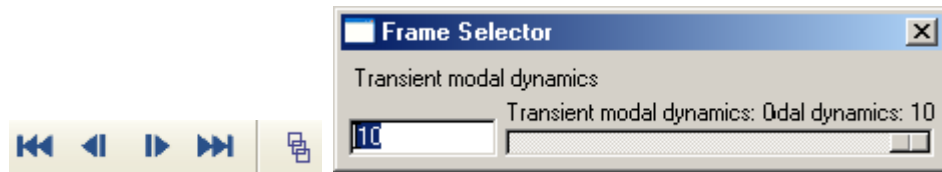
26. In the toolbox area click on the “Common Plot Options” icon
- Note that the Deformation Scale Factor can be set on the “Basic” tab
 - Click “OK”



27. In the menu bar click on Results → Active Steps/Frames
- Uncheck the Extract Frequencies step
 - Click “OK”



28. Click on the arrows on the context bar to change the time step being displayed
- Click on the three squares to bring up the frame selector slider bar



29. On the results tree, expand the History Output node and double click on the spatial displacement history created

