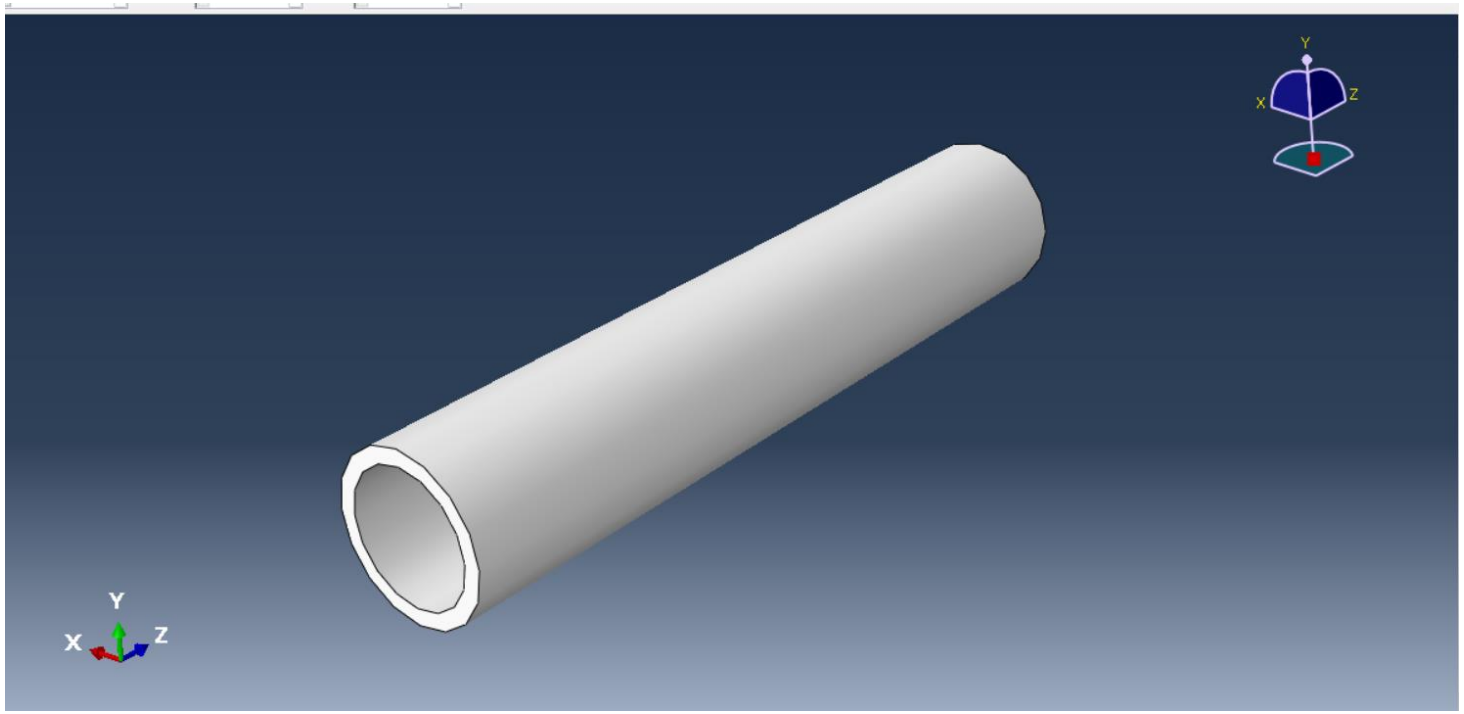
Abaqus tutorial

Modal Analysis in 3D

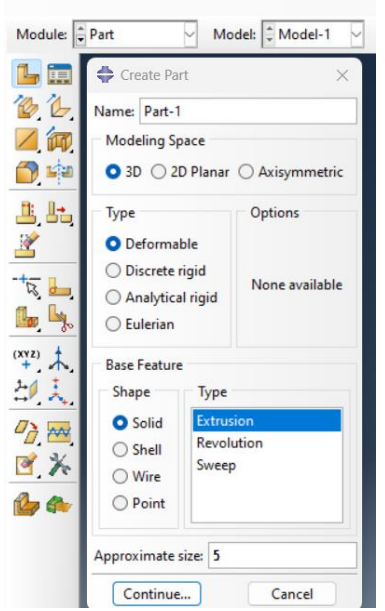
Problem

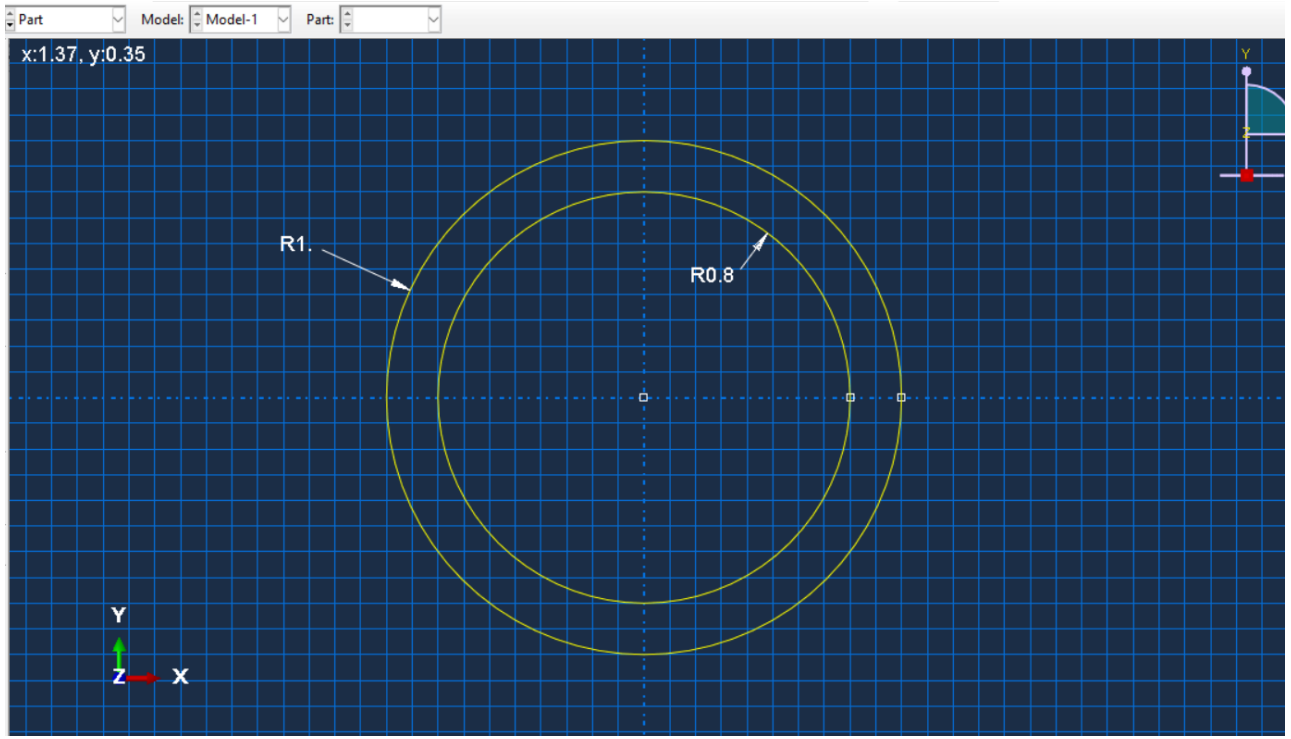
- We will be doing a modal analysis to find the first 7 natural frequencies of a 3D hollow cylinder as shown below, which has one of the ends fixed.

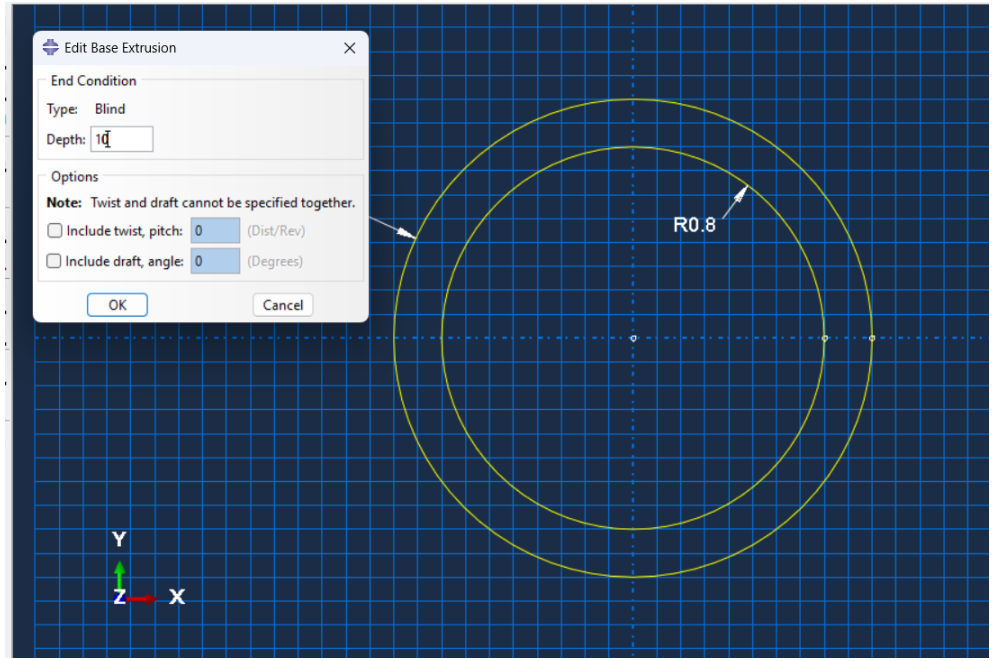


Part Module

- Click Create Part with the settings as shown in the figure.
- Create a sketch as shown below. Enter Depth 10.

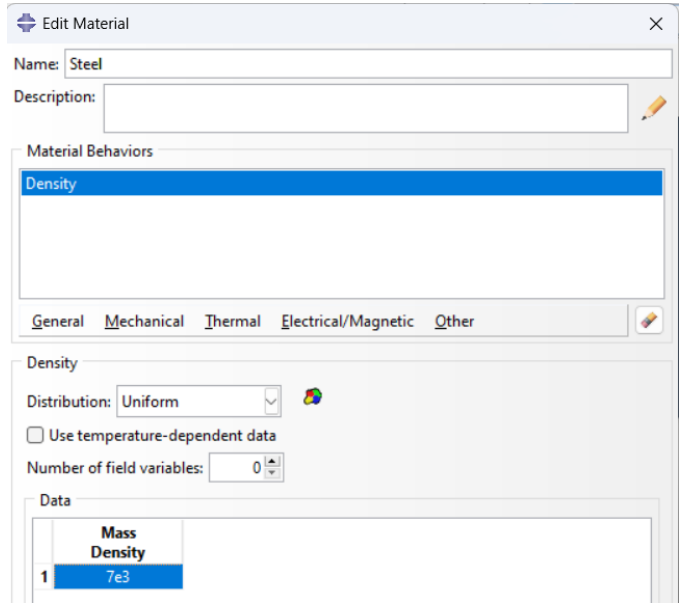


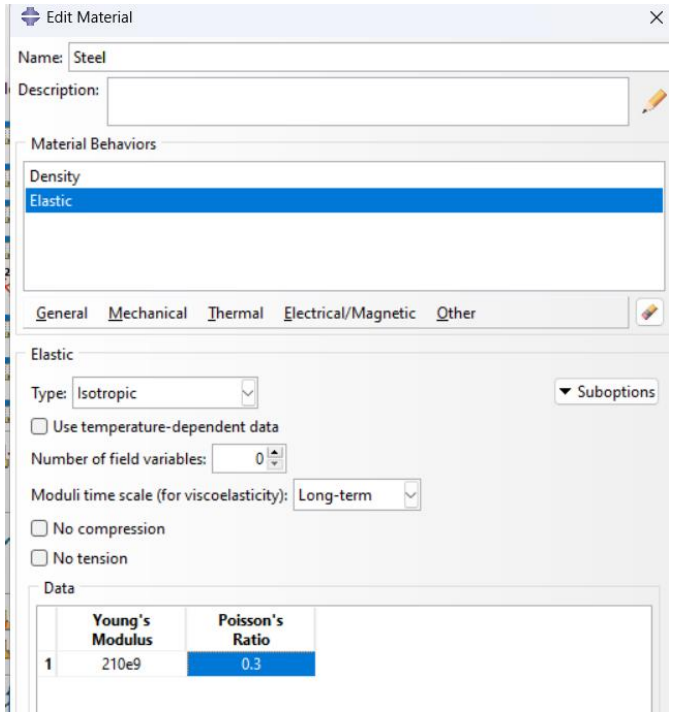




Property Module

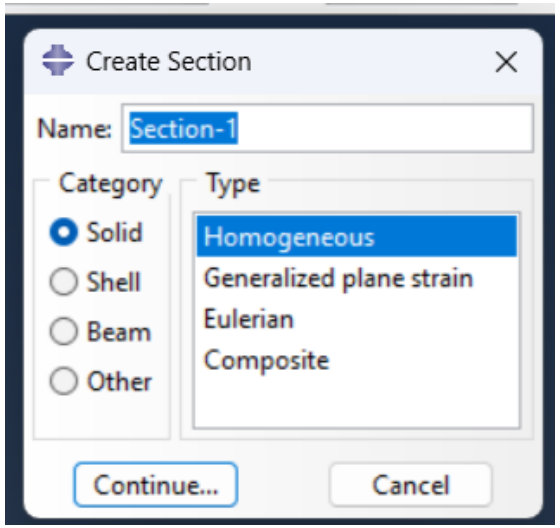
- Click on Create Material.
- Name it Steel.
- Set General -> Density = 7e3.
- Set Mechanical -> Elasticity -> Elastic, Young's Modulus = 210e9 and Poisson's Ratio = 0.3.

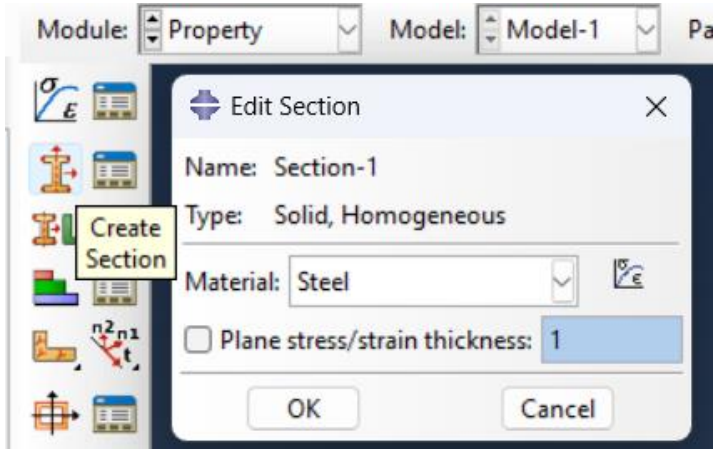


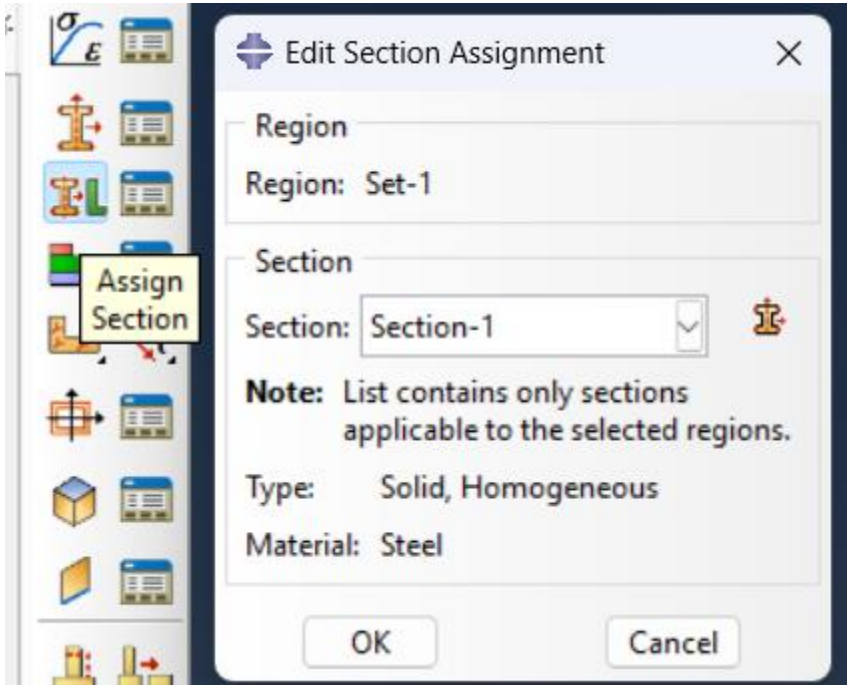


Property Module

- Click on Create Section. Select options shown on the right and Click OK.
- Next Click Assign Section. Select the section created and Click OK.

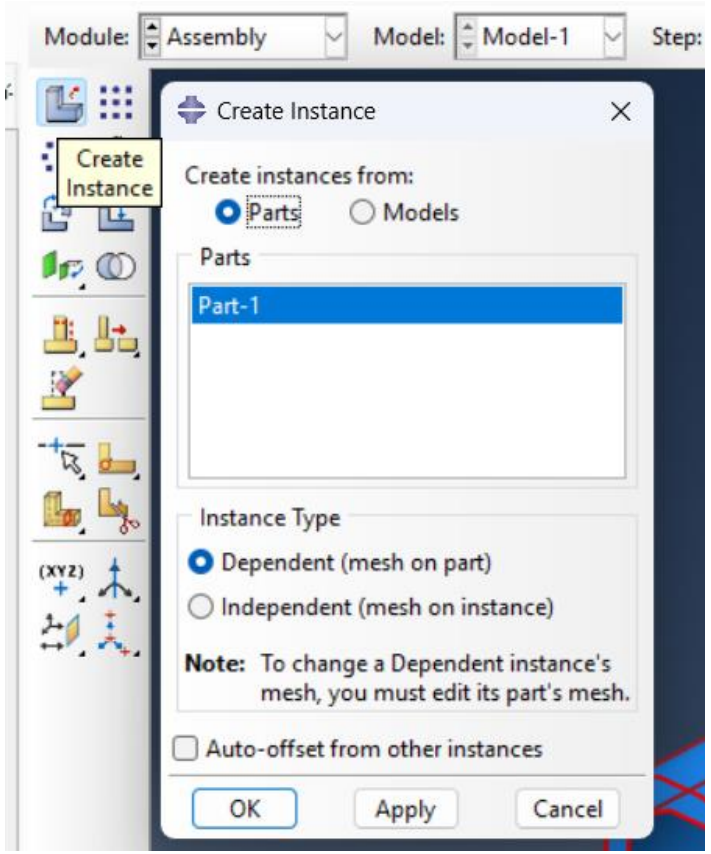






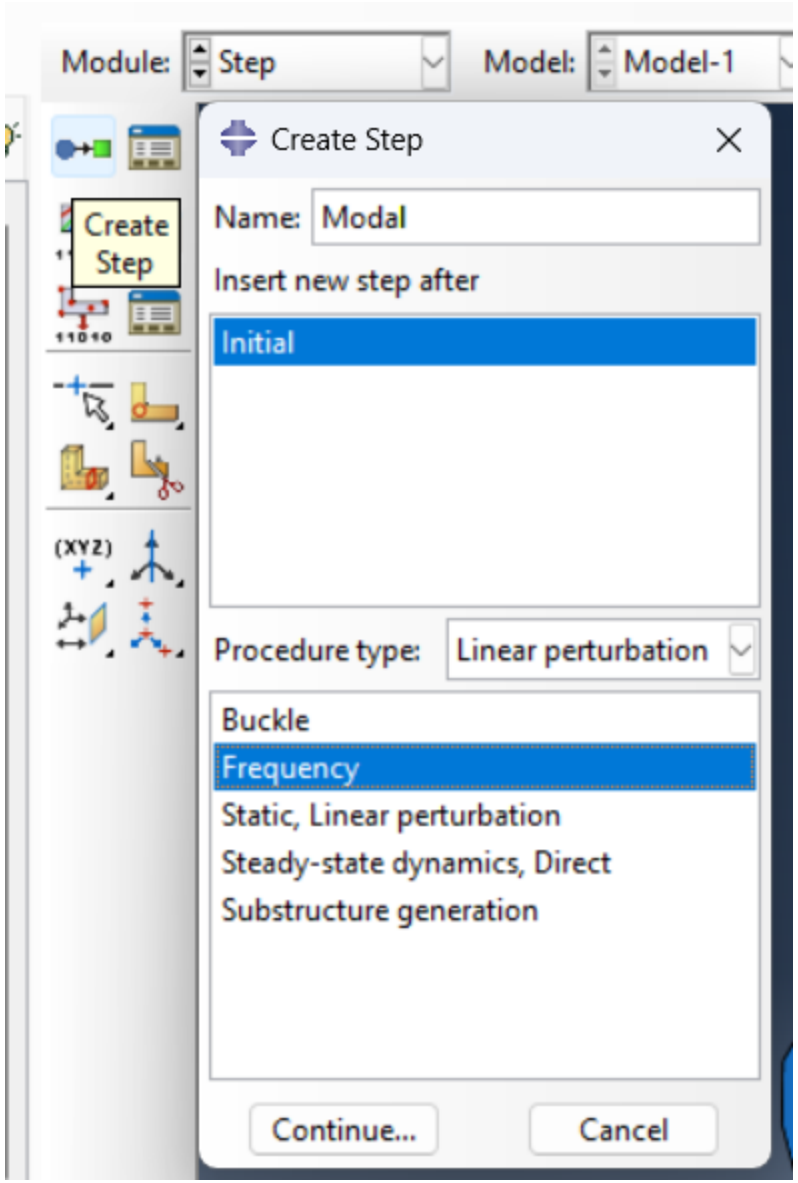
Assembly Module

- In Assembly Module, click on Create Instance, select the part, check the Dependent option and click OK.



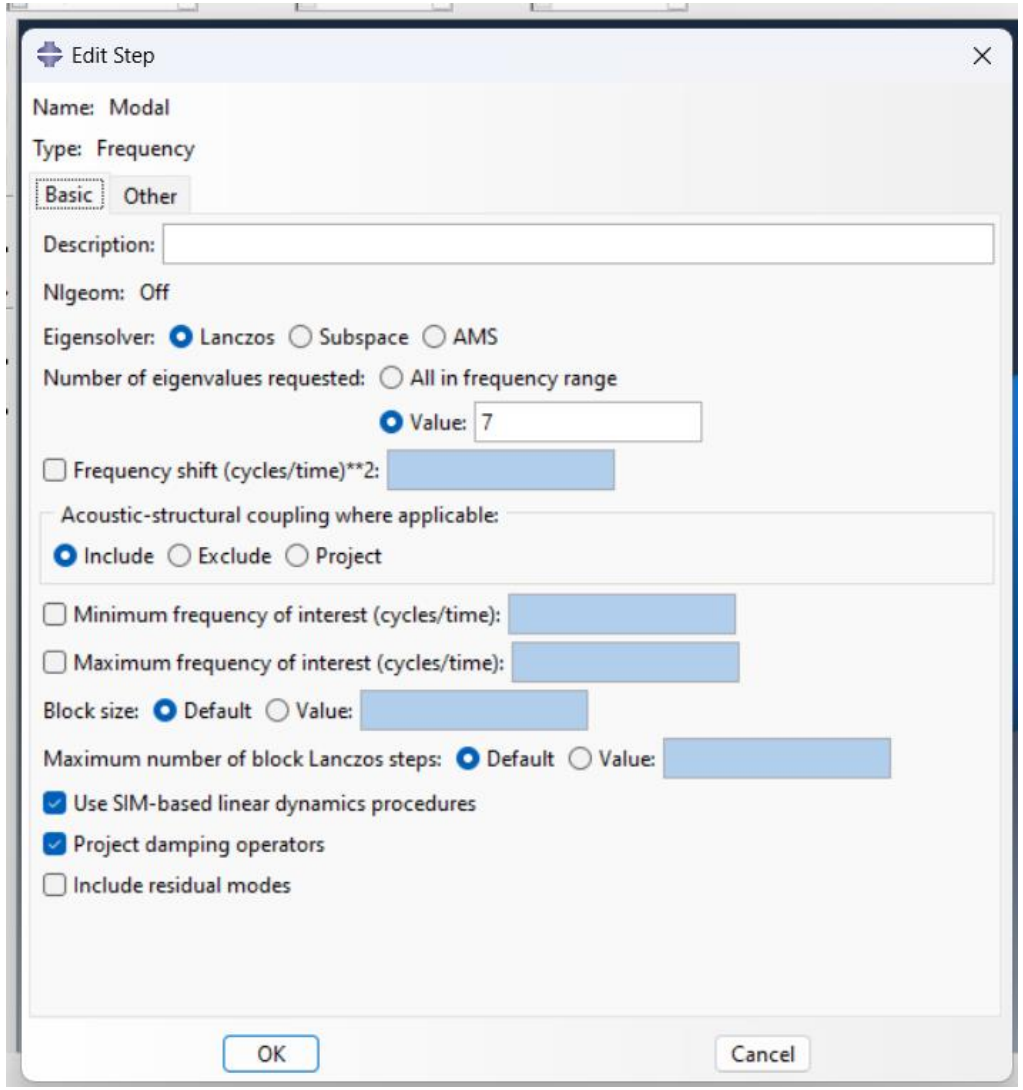
Step Module

- In Step Module, click on Create Step.
- Name it Modal.
- Select Procedure type : Linear perturbation and select Frequency.
- Click Continue



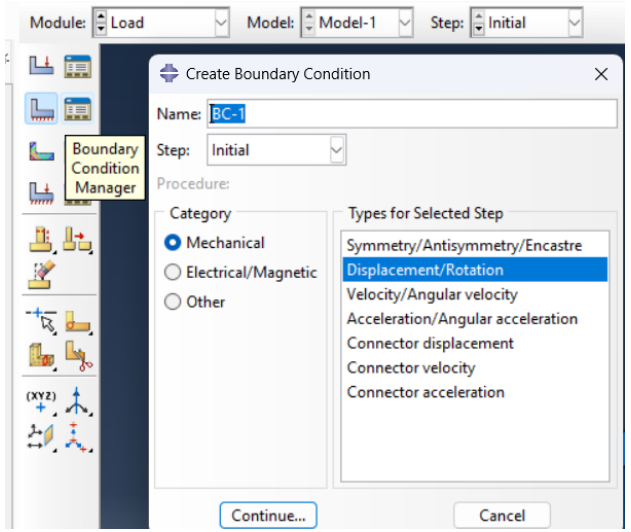
Step Module

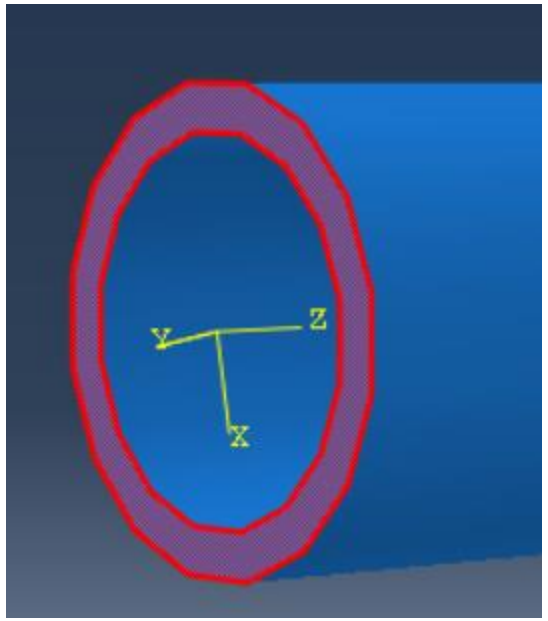
- In the next window, select Lanczos eigensolver.
- Set the “Value” of “Number of eigenvalues requested” to 7.
- Click OK

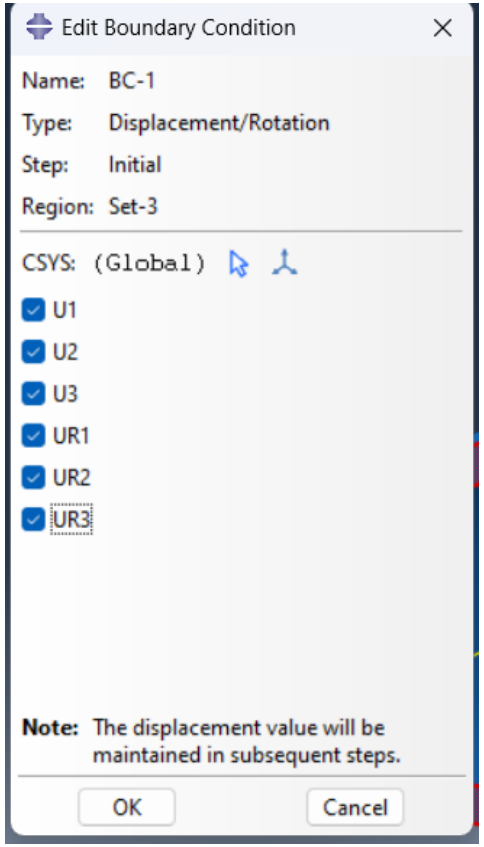


Load Module

- In load module, select “Boundary Condition Manager”.
- Select Step to Initial.
- Select Displacement/Rotation. Click Continue.
- Select a flat circular face. Click Done.
- Check all quantities in the next screen and Click OK.

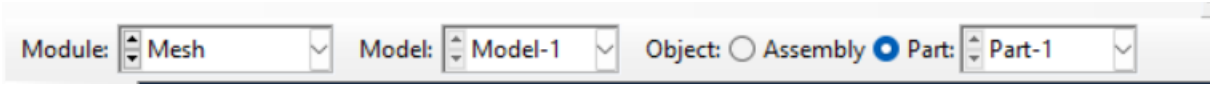


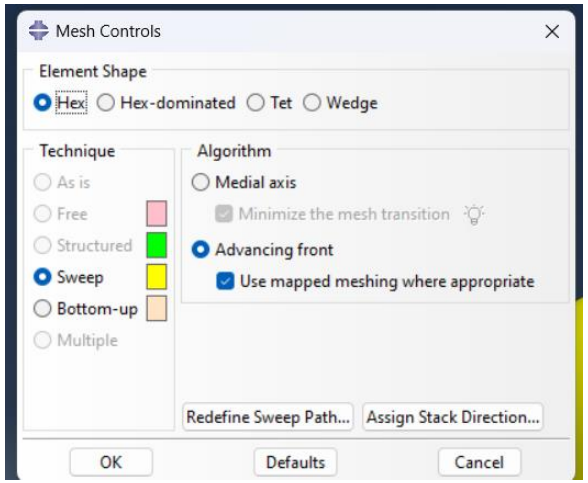


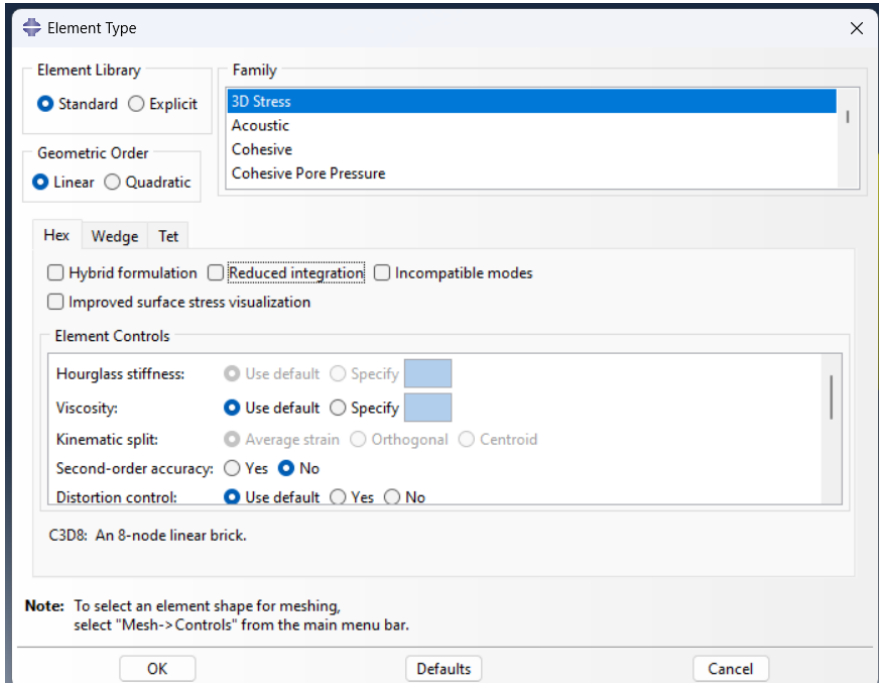


Mesh Module

- Click on Part instead of Assembly as shown on the right.
- On the toolbar Mesh -> Control and select Hex with Sweep and click OK.
- In Mesh -> Element Type, Select Hex, 3D Stress, Linear and uncheck reduced integration. Element should be C3D8.

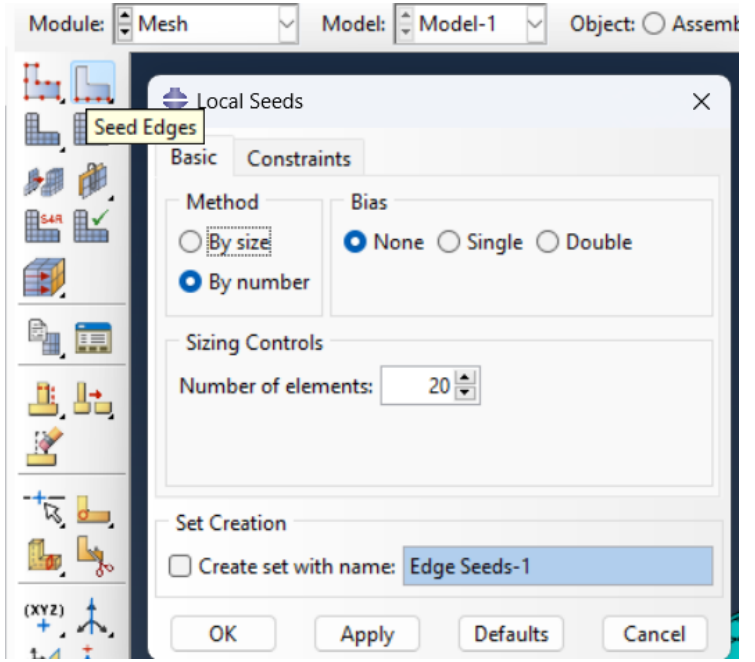


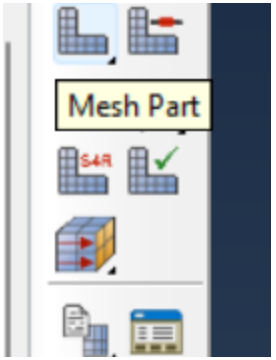


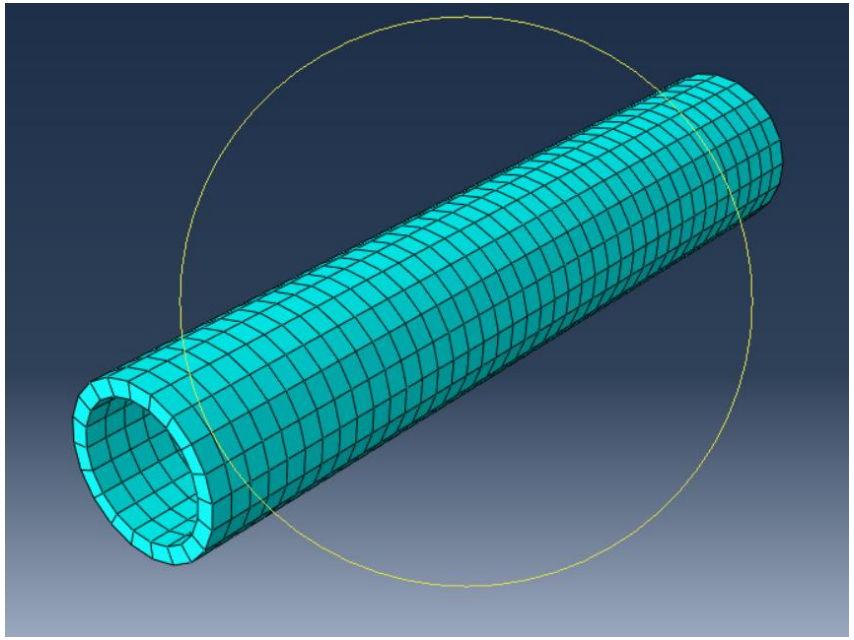


Mesh Module

- Click on Seed Edges and select the four circles on the flat faces (you should hold *Shift key* to select multiple edges and *Ctrl + Alt + Click* to rotate the body) and click Done.
- Set By number and enter Number of elements = 20.
- Click on Mesh Part and select Yes.
- You should get a mesh as shown.

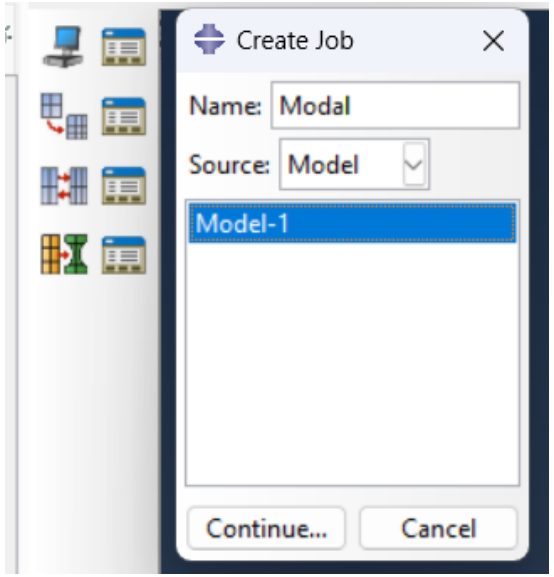


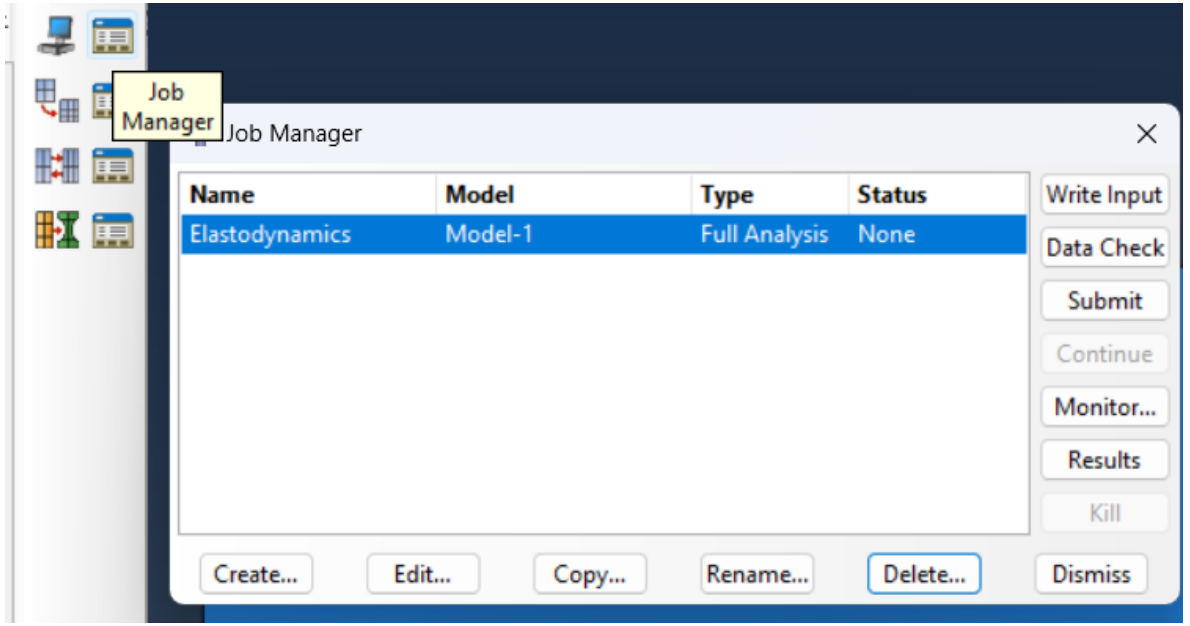




Job Module

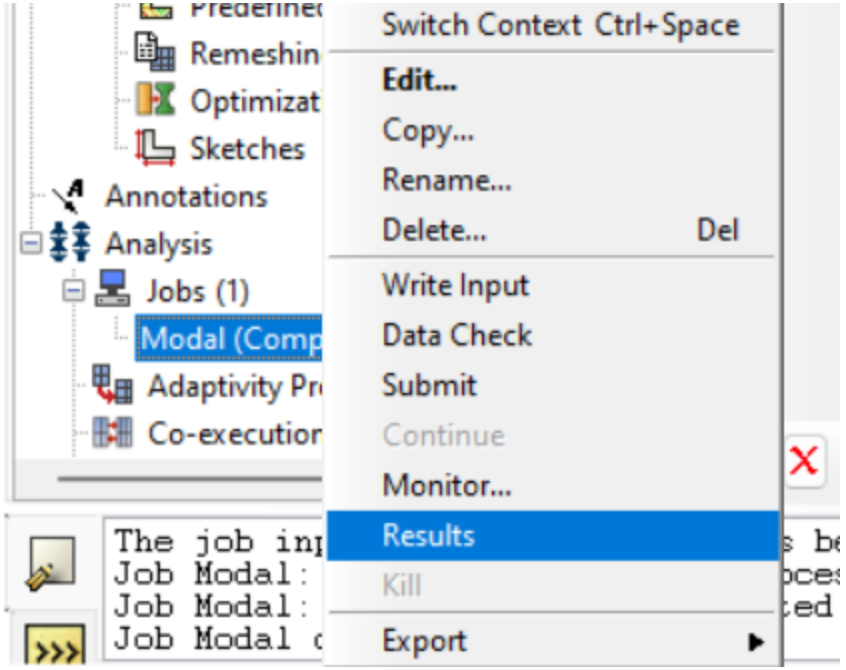
- Click on Create Job. Name it as Modal. Select the model and click Continue.
- In the next screen, click OK.
- Click on Job Manager, select Modal and click on Submit.
- The program will run for a few seconds.





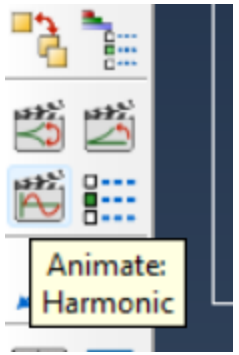
Visualization Module

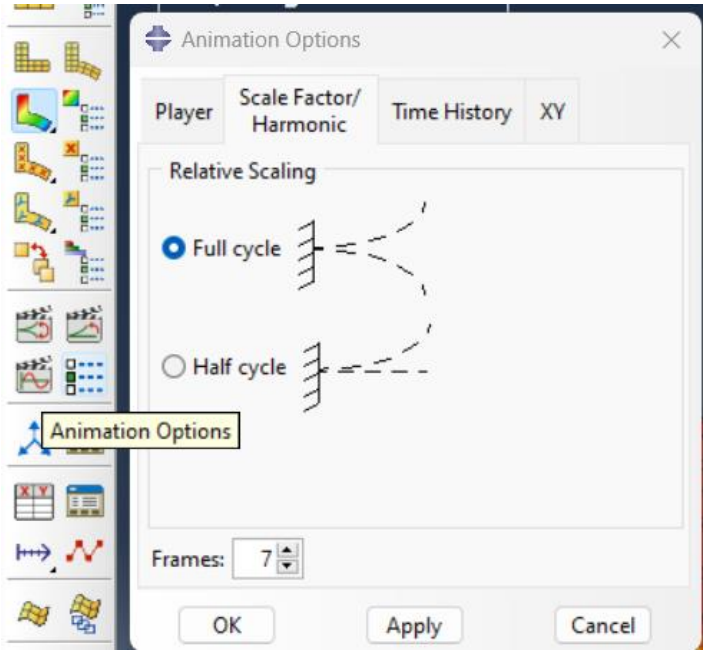
- After the job is completed, right click on the Modal job and select results.
- This will go to the results tab.

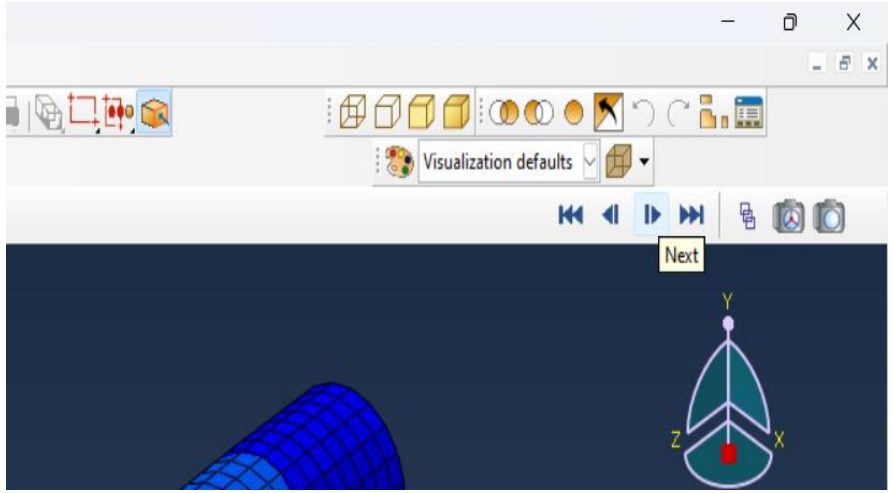


Visualization Module

- To see the current mode, Click on “Animate Harmonic”.
- To switch between the modes, stop the current animation by clicking on the “Animate Harmonic” button again.
- Select the Next or Previous buttons on the top right of the screen.
- In “Animation Options”, chose Scale Factor/ Harmonic tab and check Full Cycle.
- The animation speed can be set in Player tab of “Animation Options”.

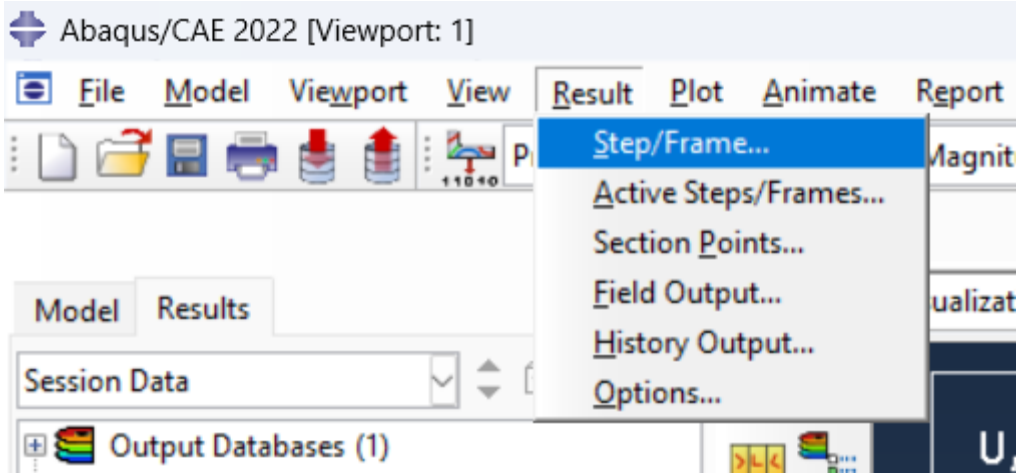


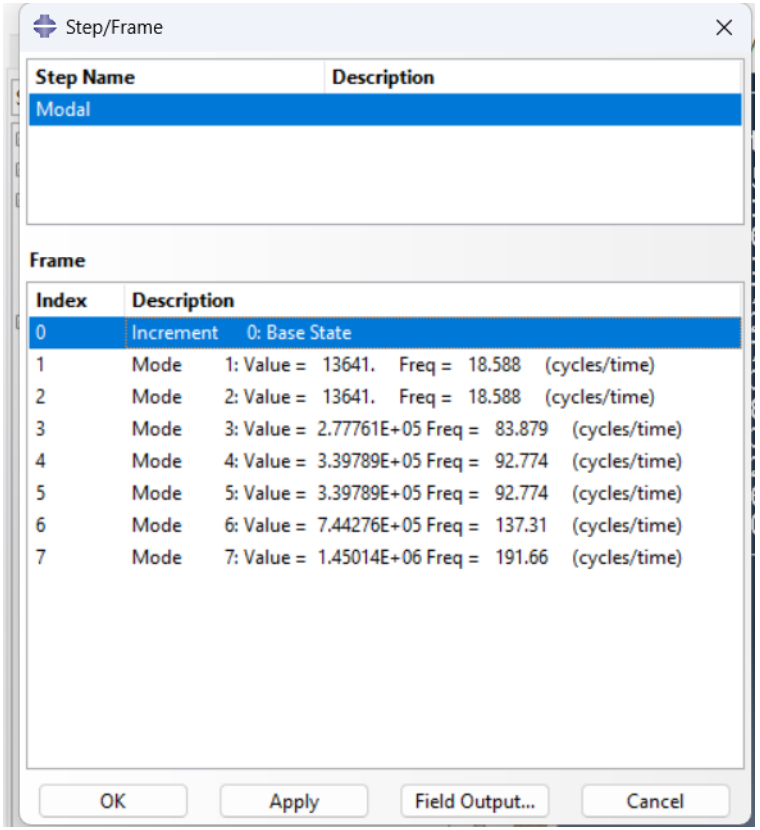




Visualization Module

- You can view the frequency of each mode as well.
- Select Result -> Step/Frame.
- A window which has the Natural frequencies will pop up.





Unconstrained Model

Make a copy of the completed model. In the copy, remove the cantilever displacement boundary condition. Create a new job associated with the new model and re-run the analysis.

When reviewing results, think about the following:

1. How do the modal frequencies differ for the unconstrained case?
2. Is there a modal frequency and deformation mode in the unconstrained case that closely matches any frequencies associated with the constrained model?