Natural Frequency Analysis in Abaqus – Step-by-Step Guide

- 1. Create a copy of the plate model:
 - In the *Model Tree*, right-click on **Model-1**, select **Copy Model**, and enter a name for the new model, e.g., **dynamics**.
 - All changes below should be made in the copied model (e.g., *dynamics*).

2. Modify the copied model:

a. Add material density:

- Go to Model Tree > Materials > Material-1 > General > Density.
- Enter the material density value under Mass Density.
 (Important: Make sure you are editing the copied model, not the original one!)

b. Set up the frequency analysis step:

i. Suppress the static step:

 Go to *Model Tree > Steps*, right-click on Step-1 (Static, General) and choose Suppress.

ii. Create a new step for modal analysis:

- In *Steps*, choose **Create Step**.
- Select *Procedure type*: Linear Perturbation > Frequency, click Continue.
- Keep default settings (Lanczos eigensolver).
- Enable the option "Number of eigenvalues requested" and set the value to 10 (this limits the number of vibration modes to be calculated).

3. Create and run the job:

- Go to *Model Tree > Jobs*, create a new job (e.g., vibration avoid using special/non-English characters).
- Submit the job and wait for the analysis to finish.

4. View the results:

- Open the *Results* module.
- To display mode shapes, select **Plot Deformed Shape**.
- You can browse through different modes using the **Frame Selector** icon in the top-right corner or through *Model Tree > Step > Frames*.
- In the plot legend:
 - Value = Eigenvalue

- **Freq** = Frequency in Hz
- To animate a vibration mode, click the **Animate: Time History** icon in the toolbar or in the *Model Tree*.

d. To compare deformed and undeformed shapes:

• Click the **Allow Multiple Plot States** icon to overlay the mode shape on the original geometry.

e. To display boundary conditions:

Go to View > ODB Display Options > Entity Display, and check Show boundary conditions.