

# ANSYS - Vibration Analysis of a Frame - Step 9

## Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
- 9. Validate the results**

## Step 9: Validate the results

Not available yet.

Back to [Problem Specification](#)

Go to [all ANSYS Learning Modules](#)