

# ANSYS - Vibration Analysis of a Frame - Step 6

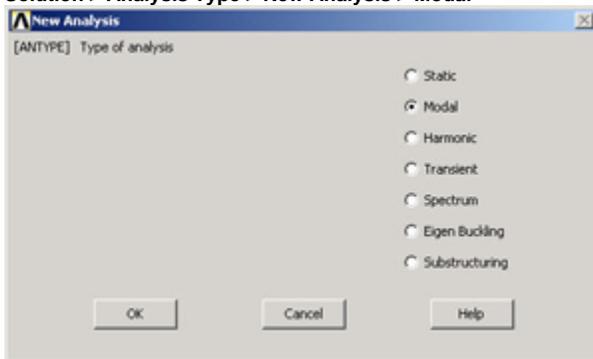
- Problem Specification
1. Start-up and preliminary set-up
  2. Specify element type and constants
  3. Specify material properties
  4. Specify geometry
  5. Mesh geometry
  - 6. Specify boundary conditions**
  7. Solve!
  8. Postprocess the results
  9. Validate the results

## Step 6: Specify boundary conditions

### Set Options

Select in *Main Menu*:

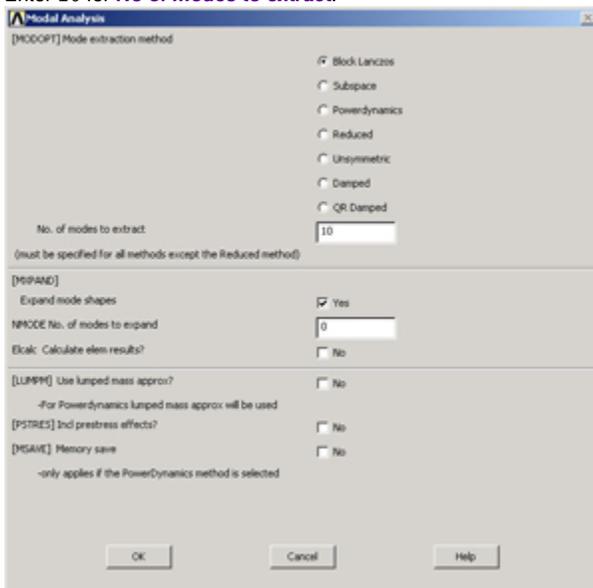
**Solution > Analysis Type > New Analysis > Modal**



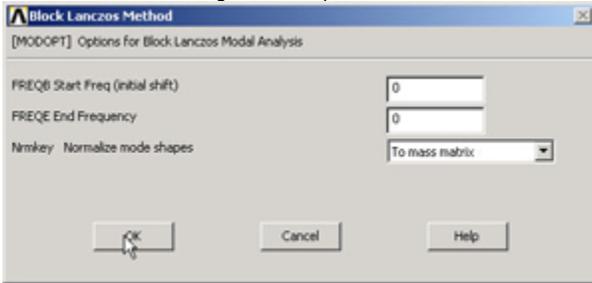
Then select in *Main Menu*:

**Solution > Analysis Type > Analysis Options**

Enter 10 for *No of modes to extract*.



Click **OK** and then **OK** again to accept defaults for the *Block Lanczos Method*.

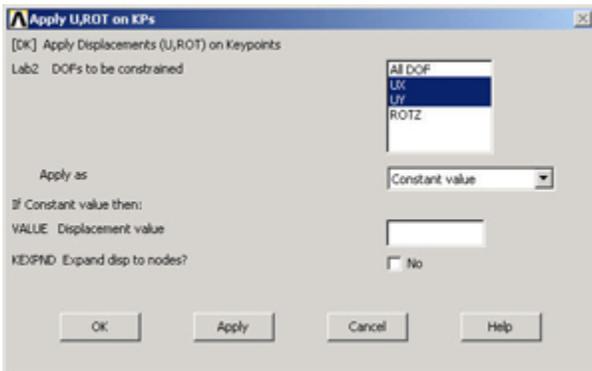


## Apply Displacement Constraints

Select in *Preprocessor*:

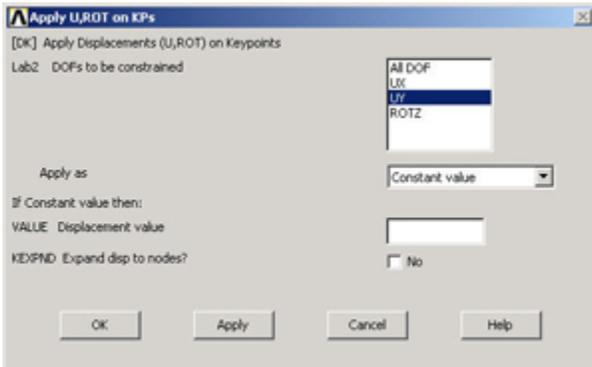
**Loads > Define Loads > Apply > Structural > Displacement > On Keypoints**

Select keypoint at A. Select **UX** and **UY**, Enter 0 for *Displacement value*.

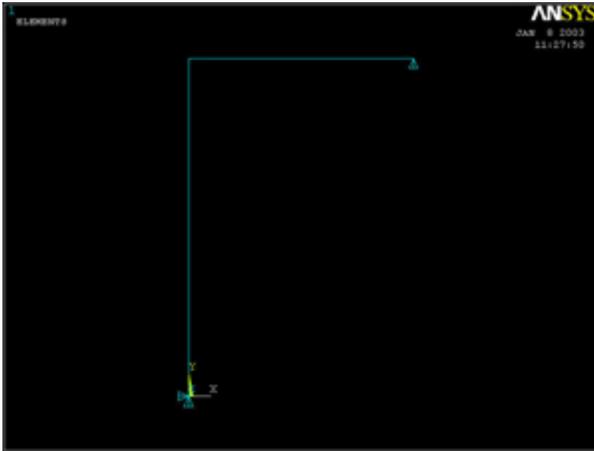


Click **OK**.

Select keypoint at C. Select **UY**, Enter 0 for *Displacement value*.



Click **OK**.

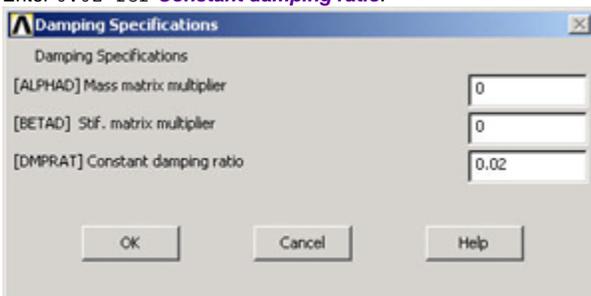


## Specify Damping Ratio

Select in *Preprocessor*:

**Loads > Load Step Opts > Time/Frequency > Damping**

Enter 0.02 for **Constant damping ratio**.



Click **OK**.

## Save your work

Click on **SAVE\_DB** in the *ANSYS Toolbar* to save the database.

Go to [Step 7: Solve!](#)

Go to [all ANSYS Learning Modules](#)