

ANSYS - Vibration Analysis of a Frame - Step 6

Problem Specification

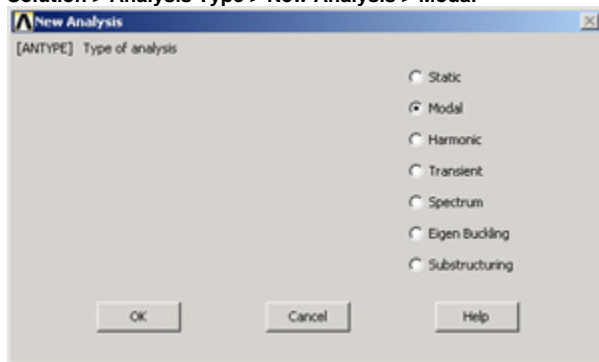
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
- 6. Specify boundary conditions**
7. Solve!
8. Postprocess the results
9. Validate the results

Step 6: Specify boundary conditions

Set Options

Select in *Main Menu*:

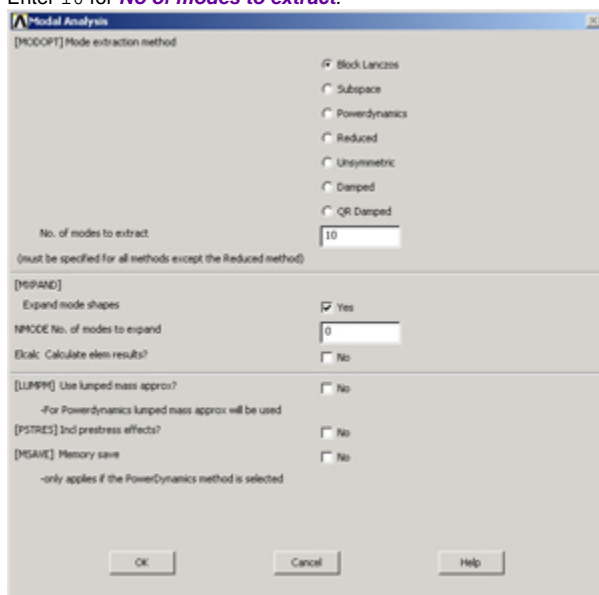
Solution > Analysis Type > New Analysis > Modal



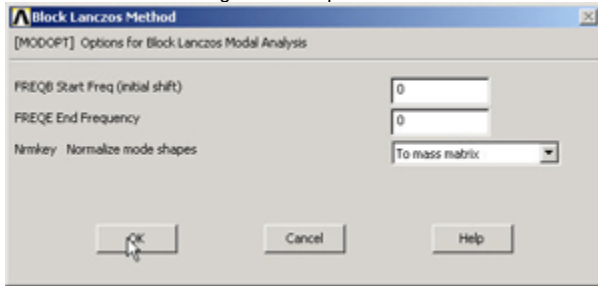
Then select in *Main Menu*:

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*.

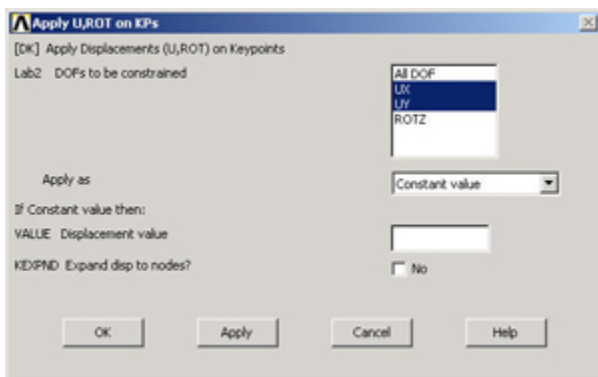


Apply Displacement Constraints

Select in *Preprocessor*:

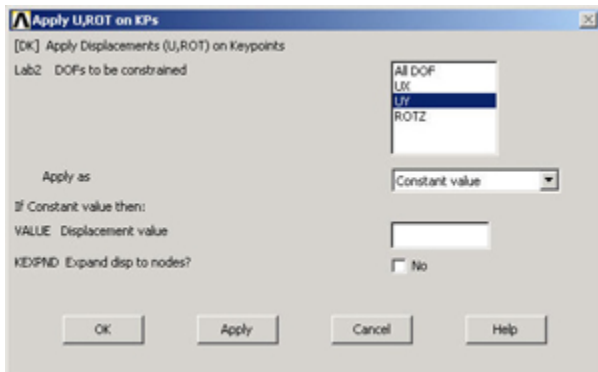
Loads > Define Loads > Apply > Structural > Displacement > On Keypoints

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

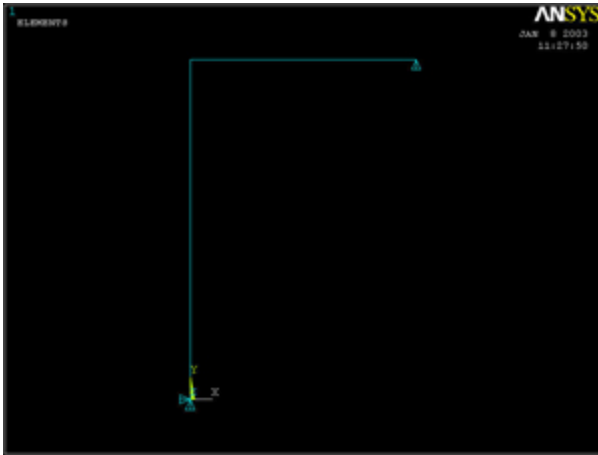


Click **OK**.

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



Click **OK**.

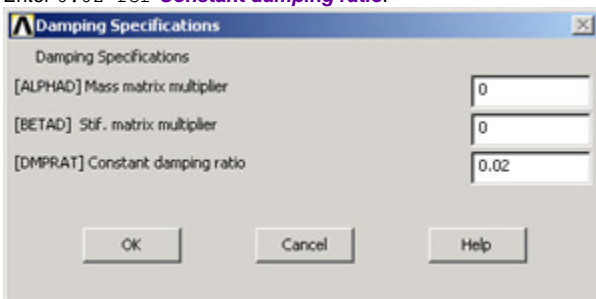


Specify Damping Ratio

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](#)