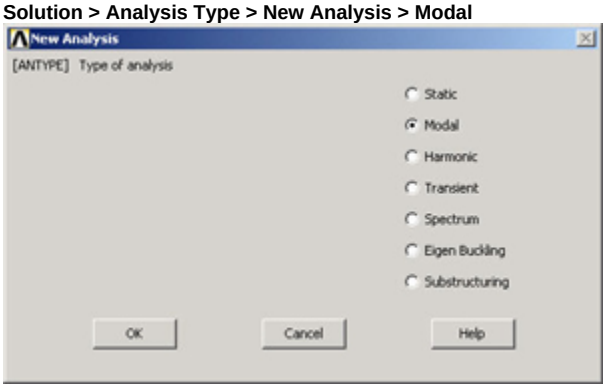
ANSYS - Vibration Analysis of a Frame - Step 6

Step 6: Specify boundary conditions

Set Options

Select in *Main Menu*:

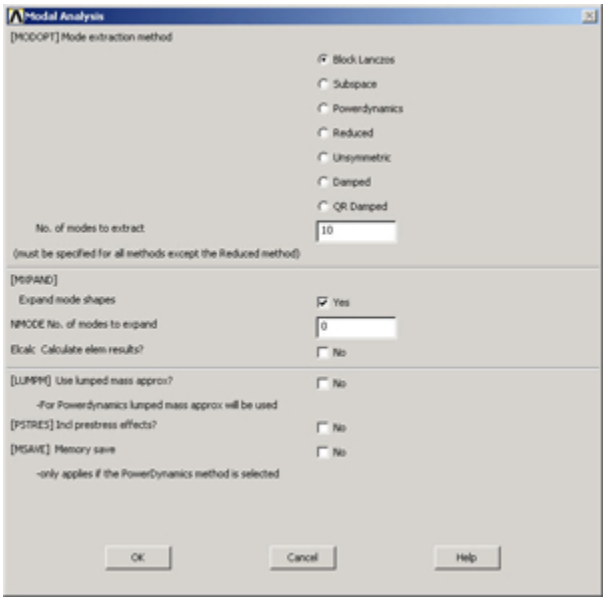
Solution > Analysis Type > New Analysis > Modal



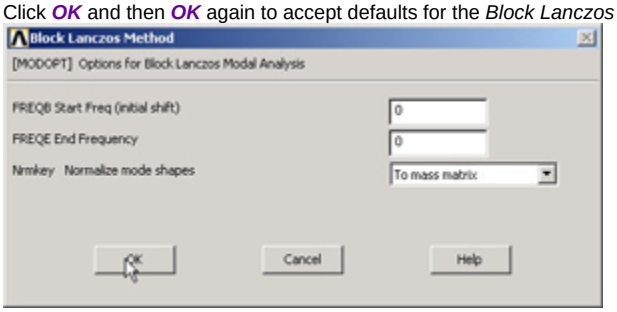
Then select in *Main Menu*:

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*.

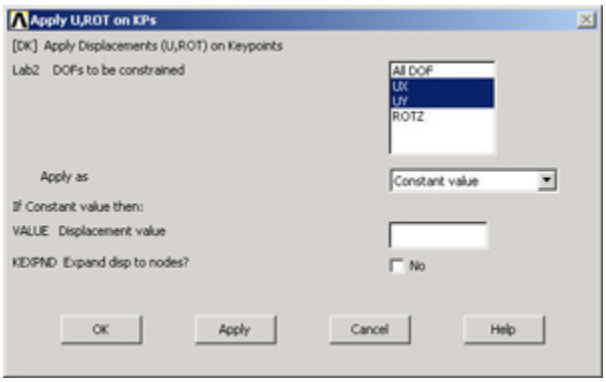


Apply Displacement Constraints

Select in *Preprocessor*:

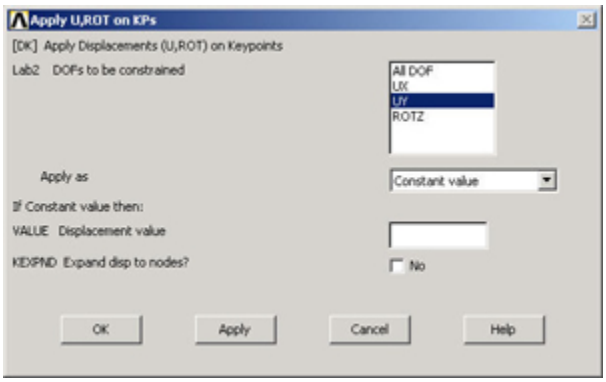
Loads > Define Loads > Apply > Structural > Displacement > On Keypoints

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

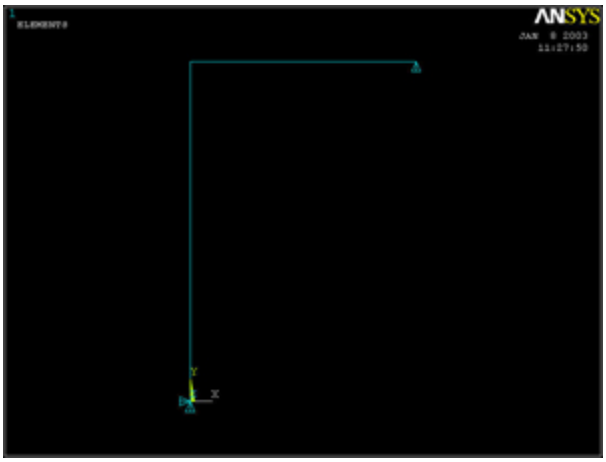


Click ***OK***.

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



Click ***OK***.

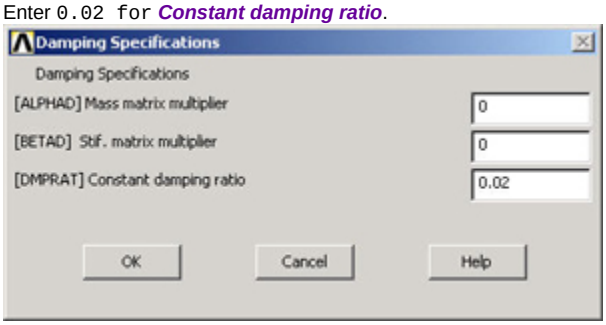


Specify Damping Ratio

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