- **Problem Specification**
- 1. Start-up and preliminary set-up
- 2. Specify element type and constants
- 3. Specify material properties
- Specify geometry
 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

New Analysis	<u>×</u>
[ANTYPE] Type of analysis	
	C Static
	Modal
	C Harmonic
	C Transient
	C Spectrum
	C Eigen Buckling
	C Substructuring
OK Cancel	Help

Then select in Main Menu:

Solution > Analysis Type > Analysis Options

Enter 10 for No of modes to extract.

Modal Analysis	
[MODOPT] Mode extraction method	
	@ Block Lanczos
	C Subspace
	C Powerdynamics
	C Reduced
	C Unsymmetric
	C Damped
	C OR Damoed
No. of modes to extract	10
(must be specified for all methods except the Reduced method	1 m
Descured	·
[Potward mode change	5.00
Expand mode snapes	12 Yes
NMODE No. of modes to expand	0
Elcalc Calculate elem results?	□ No
[LUMPH] Use lunped mass approx?	LE No
-For Powerdynamics lumped mass approx will be used	
[PSTRES] Incl prestress effects?	E No
[HSAVE] Hemory save	E No
-only applies if the PowerDynamics method is selected	
OK C	ancel Help

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.

[DK] Apply Displacements (U,ROT) on Keypoints	
ab2 DOFs to be constrained	All DOF UK UK ROTZ
Apply as	Constant value
f Constant value then:	
ALUE Displacement value	
(DIPND Expand disp to nodes?	IT No
OK Apply	Cancel Help

Click OK.

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

Apply U,ROT on KPs	×
[DK] Apply Displacements (U,ROT) on Keypoints	
Lab2 DOFs to be constrained	Al DOF UX VY ROTZ
Apply as	Constant value
If Constant value then:	
VALUE Displacement value	
KERPND Expand disp to nodes?	IT No
OK Apply	Cancel Help





Specify Damping Ratio

Select in Preprocessor.

Loads > Load Step Opts > Time/Frequency > Damping

Enter 0.02 for Constant damping ratio.

is	×
lier	0
ar -	0
ratio	0.02
Cancel	Help
	ier er prabo Cancel

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