

ANSYS - Vibration Analysis of a Frame - Step 4

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 4: Specify geometry

Create Keypoints

Select in *Preprocessor* menu:

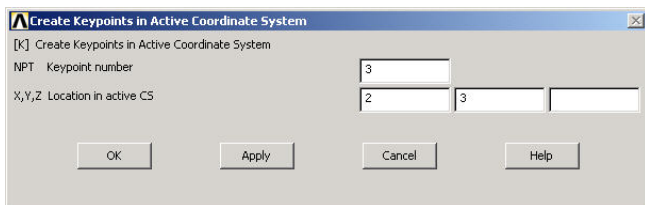
Modeling > Create > Keypoints > In Active CS

Enter:

Keypoint 1: X=0, Y=0

Keypoint 2: X=0, Y=3

Keypoint 3: X=2, Y=3



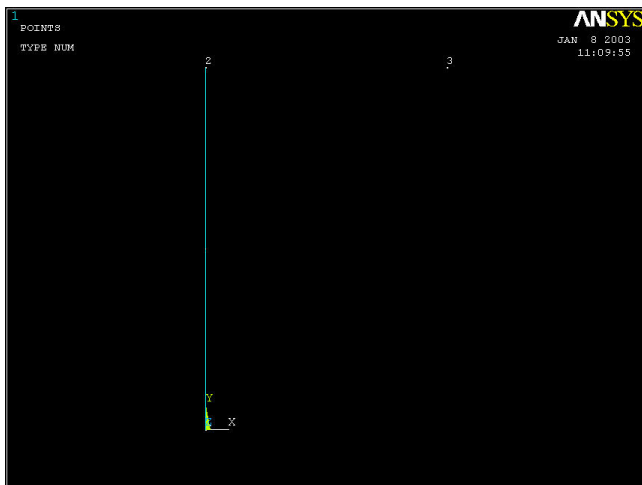
Click **OK**.

Create the Lines AB and BC

Select in *Preprocessor* menu:

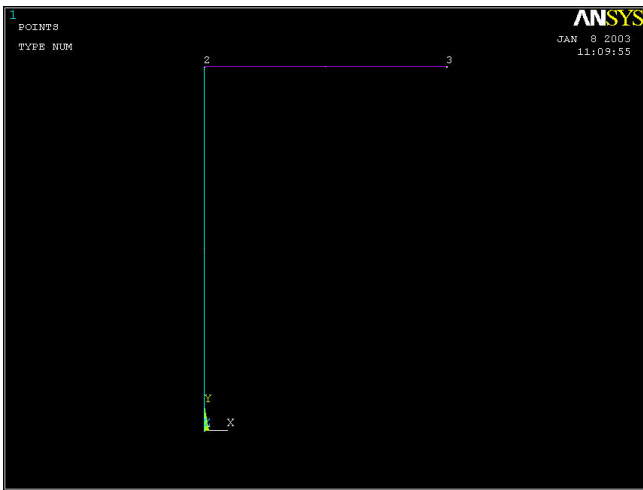
Modeling > Create> Lines > Lines > In Active Coord

Select keypoint 1 followed by keypoint 2.



Click **OK**.

Select keypoint 2 followed by keypoint 3.



Click **OK**.

Save your work

Click on **SAVE_DB** in the *ANSYS Toolbar* to save the database.

Go to [Step 5: Mesh geometry](#)

Go to [all ANSYS Learning Modules](#)