# ANSYS - Vibration Analysis of a Frame

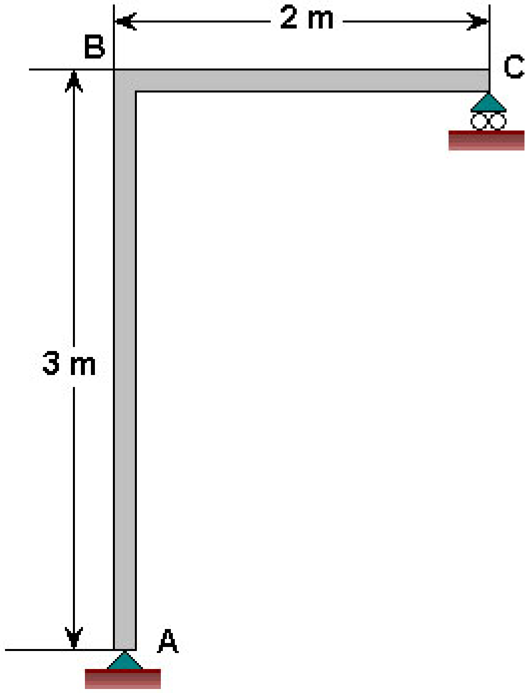



Author: Rajesh Bhaskaran, Cornell University




## Problem Specification

The problem considered here is the vibration analysis of the right-angle frame in example 11.17 on page 436 of Cook et al.




$E = 200x10^9\ Pa$

$\nu = 0.29$

$\rho = 7860 kg/m^3$

$I = \frac{1x10^{-4}}{12}$

Go to Step 1: Start-up and preliminary set-up

Go to all ANSYS Learning Modules



# Step 1: Start-up and preliminary set-up

### Create a folder

Create a folder called *dynamics* at convenient location. We'll use this folder to store files created during the session.

### Start ANSYS

**Start > Programs > ANSYS Release 7.0 > ANSYS Interactive**

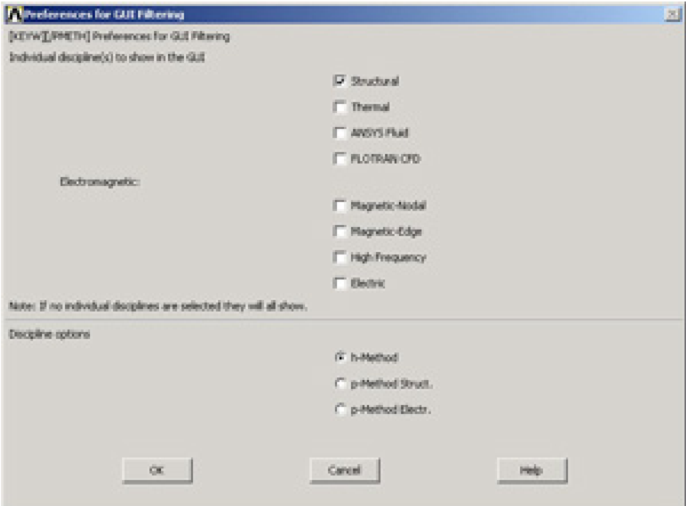
### Specify directory and job name

In the window *ANSYS Interactive 7.0 Launcher* that pops up, enter the location of the folder you just created as your ***Working directory*** by browsing to it (for example, C:\dynamics). Specify `raf` as your ***Initial jobname***. The jobname is the prefix used for all files generated by the ANSYS run. Click on *Run.*

### Set Preferences

**Main Menu > Preferences**

In the *Preferences for GUI Filtering* dialog box, click on the box next to ***Structural*** so that a tick mark appears in the box.




Recall that this is an optional step that customizes the graphical user interface so that only the menu option valid for the structural problems are made available.

### Enter Parameters

For convenience, we'll create scalar parameters corresponding to *v, I , p,* and *E.*

**Utility Menu > Parameters > Scalar Parameters**

Enter the parameter values and click ***Accept*** after each.

E = 200e9
nu = 0.29
rho = 7860
I = (1e-4)/12

Close the *Scalar Parameters* window.

We can now enter these variable names instead of the corresponding values as we set up the problem in ANSYS. This is also helpful in carrying out parametric studies.

Go to Step 2: Specify Element Types and Constants

Go to all ANSYS Learning Modules



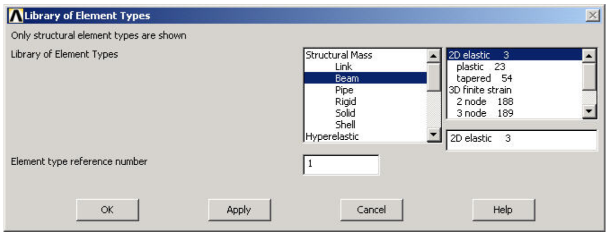
## Step 2: Specify element type and constants

### Specify Element Type

In the *Preprocessor* Menu, Select:

**Element Type > Add/Edit/Delete > Add...**

Pick ***Beam*** in the left field and ***2D elastic 3*** in the right field.




Click ***OK***.

Close the *Element Types* dialog box and also the *Element Type* menu.

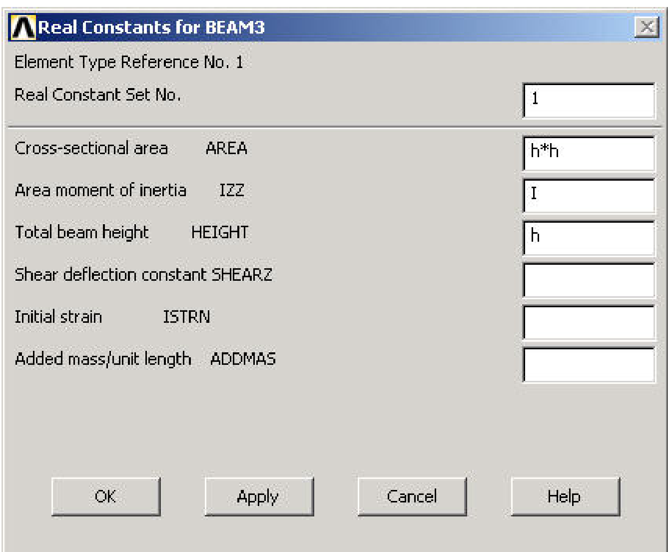
### Specify the Constants

In the *Preprocessor* menu, Select*:*

**Real Constants > Add/Edit/Delete > Add...**

This brings up the *Element Type for Real Constants* dialog box with a list of the element types defined in the previous step. Click ***OK*** to select the ***BEAM3*** element. Enter the following values:

*AREA* = h*h
*IZZ* = I
*HEIGHT* = h




Save your work by clicking on the ***Save_DB*** button in the *ANSYS Toolbar.*

Go to Step 3: Specify material properties

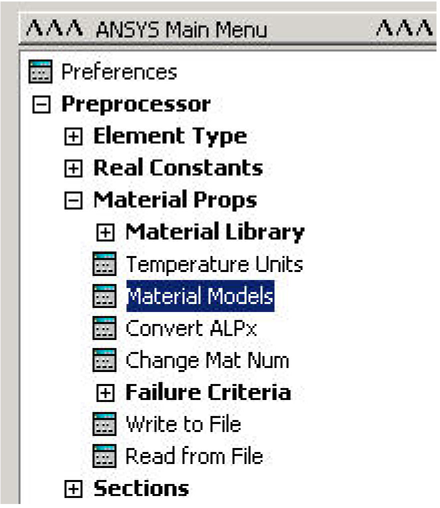
Go to all ANSYS Learning Modules



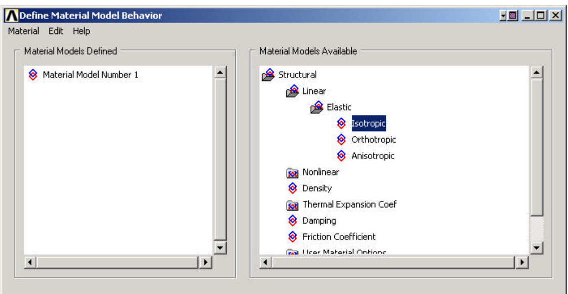
## Step 3: Specify material properties

### Enter the Define Material Model Behavior menu

**Select Main Menu > Preprocessor > Material Props > Material Models**

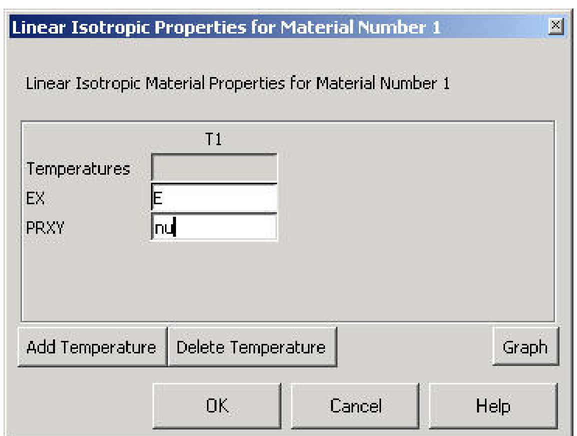



In the *Define Material Model Behavior* menu, double-click on ***Structural*, *Linear*, *Elastic*, *and Isotropic***.



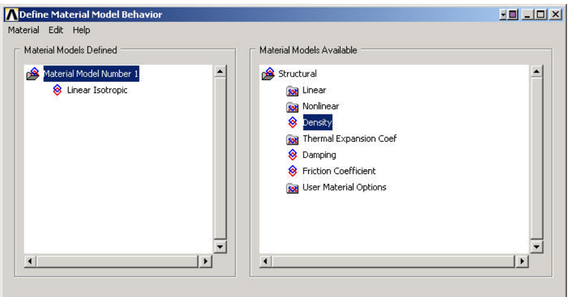

### Specify Material properties

Enter E for Young's modulus ***EX***, nu for Poisson's Ratio ***PRXY***.

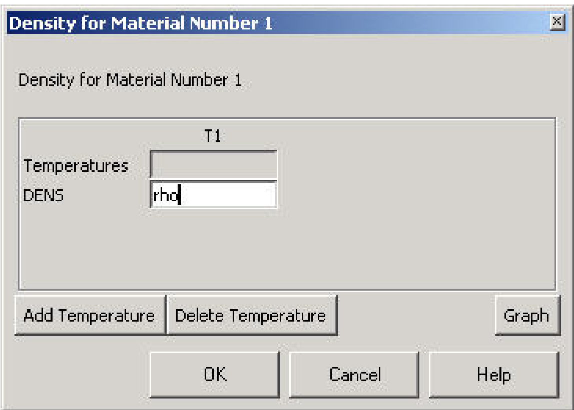



Click ***OK***.

Double-click on ***Density*** under ***Structural***.




Enter `rho` for ***DENS***.




Click ***OK***.

This completes the specification for Material Model #1. Close the *Define Material Model Behavior* menu.

### Save your work

Click on the ***SAVE_DB*** button in the *ANSYS Toolbar*.

Go to Step 4: Specify geometry

Go to all ANSYS Learning Modules



## Step 4: Specify geometry

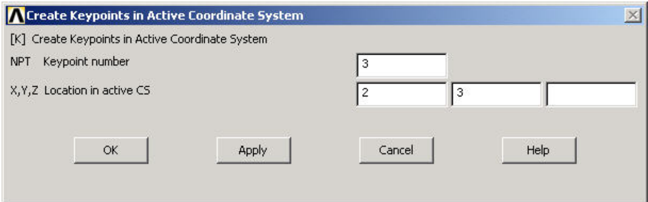
### Create Keypoints

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



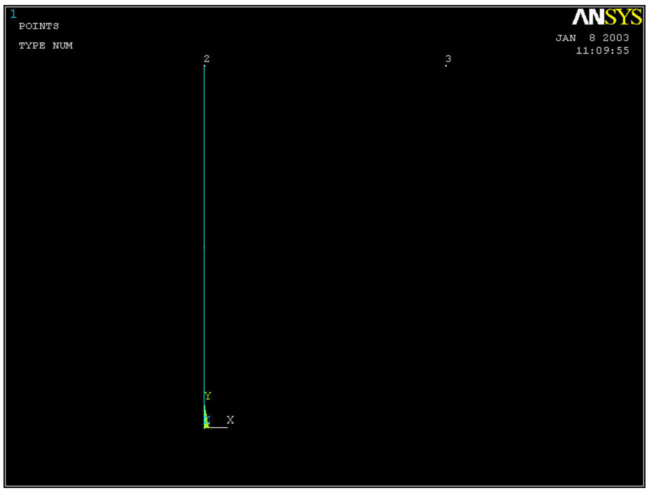

Click ***OK***.

### Create the Lines AB and BC

Select in *Preprocessor* menu:

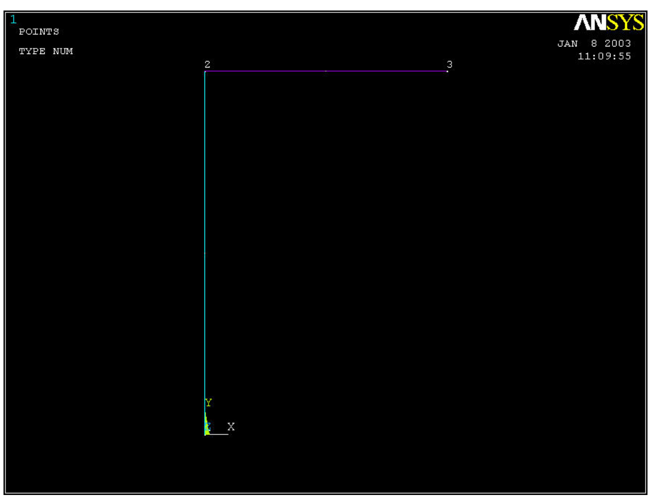
**Modeling > Create> Lines > Lines > In Active Coord**

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

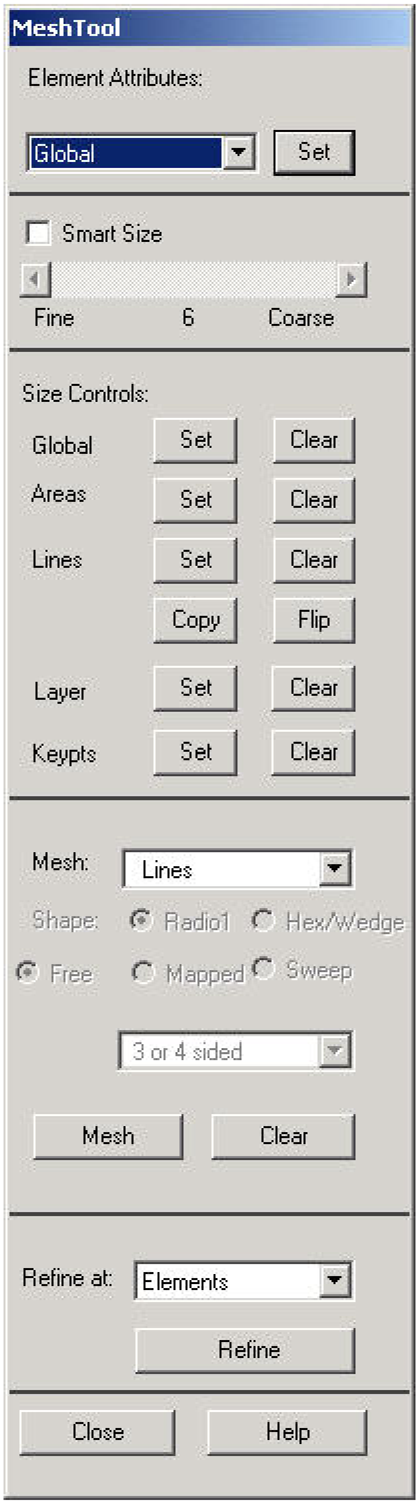
# Step 5: Mesh geometry

### Generate the Mesh

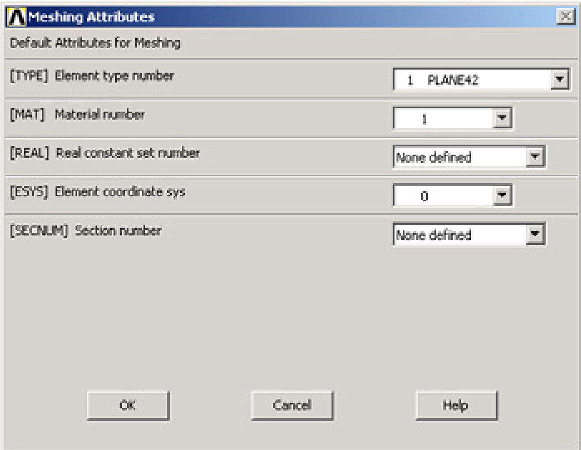
To bring up the MeshTool, Select:

**Main Menu > Preprocessor > MeshTool**.

Click on ***Set*** under ***Element Attributes*** in the MeshTool.




This brings up the *Meshing Attributes* menu. You will see that the correct element type, material number and real constants are already selected since we have only one of each.

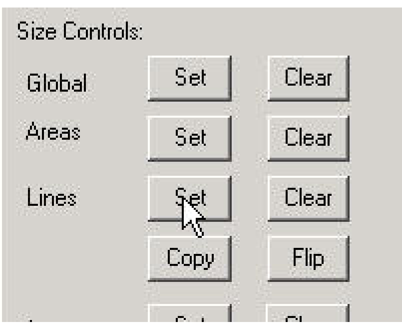



Close this menu by clicking ***OK***.

### Define Number of Elements for Each Line

We'll use 20 elements for AB and 20 elements for BC to be consistent with Cook et al.

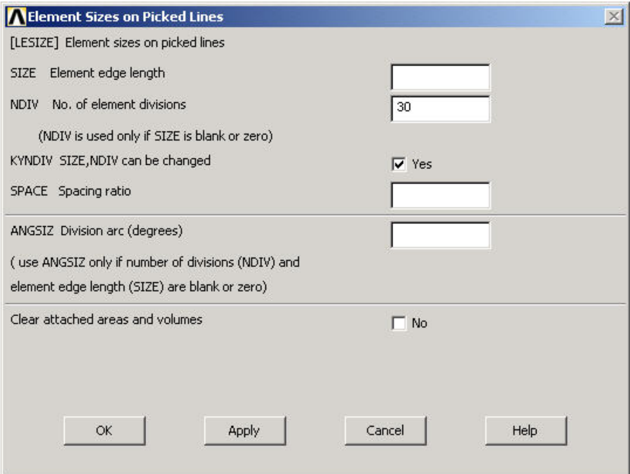
Under ***Size Control*** and ***Lines***, click ***Set***.




Select line AB.


Click ***OK***.

Enter `30` for ***NDIV***.

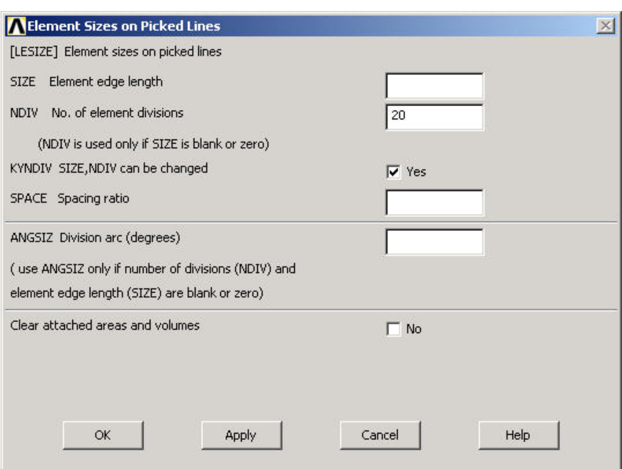



Click ***Apply***.

Select line BC.


Click ***OK***.

Enter `20` for ***NDIV***.




Click ***OK***.

### Creating the Mesh

In the *MeshTool*, click on ***Mesh***. This brings up the *pick* menu. Click on ***Pick All***.

The geometry has been meshed and the elements are plotted in the graphics window. Close the *MeshTool*.

### Save your work

Once you have successfully created the mesh, click on ***SAVE_DB*** in the *ANSYS Toolbar* to save the database.

Go to Step 6: Specify boundary conditions

Go to all ANSYS Learning Modules

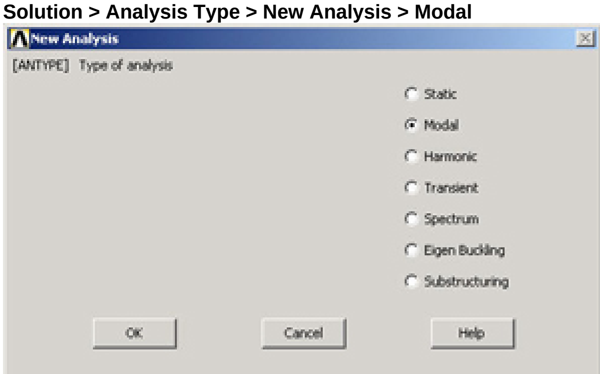


## Step 6: Specify boundary conditions

### Set Options

Select in *Main Menu*:

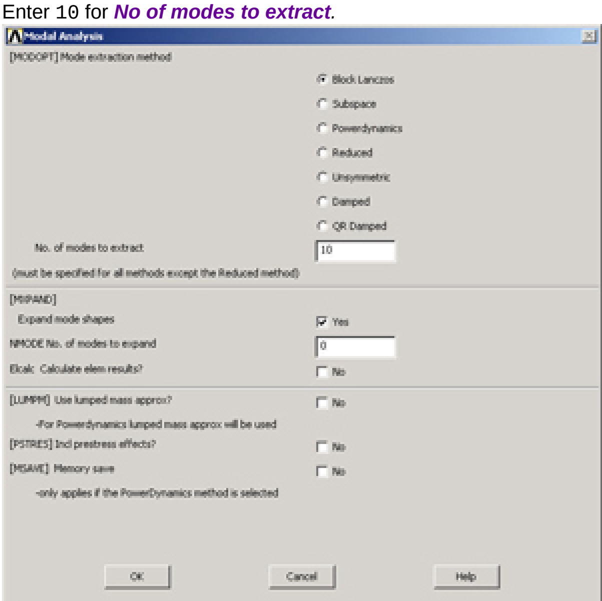
**Solution > Analysis Type > New Analysis > Modal**



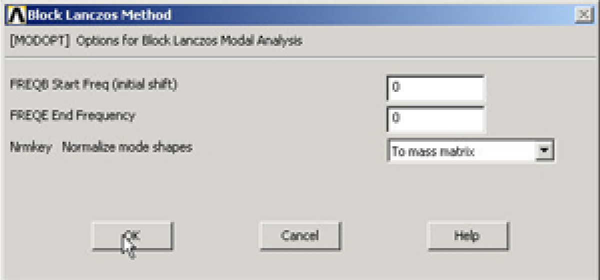

Then select in *Main Menu*:

**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*.

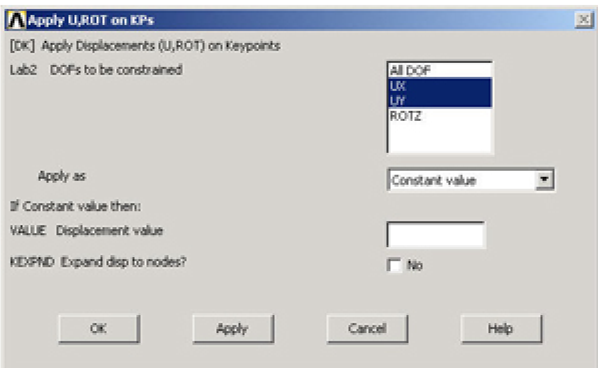



### Apply Displacement Constraints

Select in *Preprocessor*:

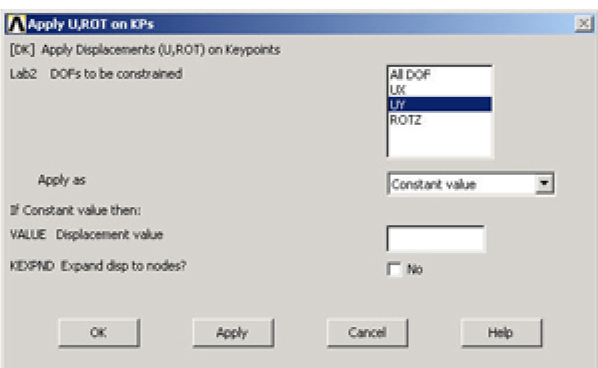
**Loads > Define Loads > Apply > Structural > Displacement > On Keypoints**

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

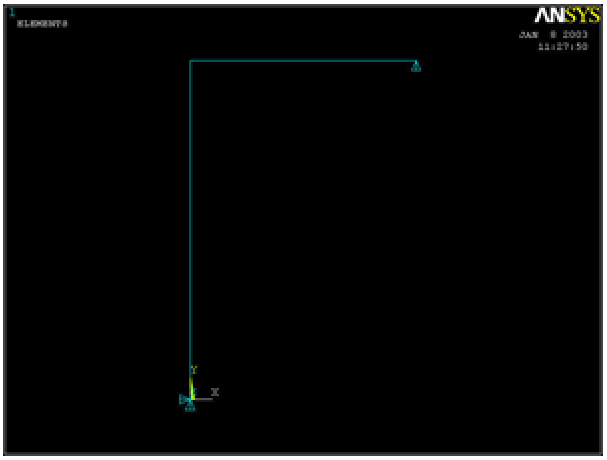



Click ***OK***.

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




Click ***OK***.

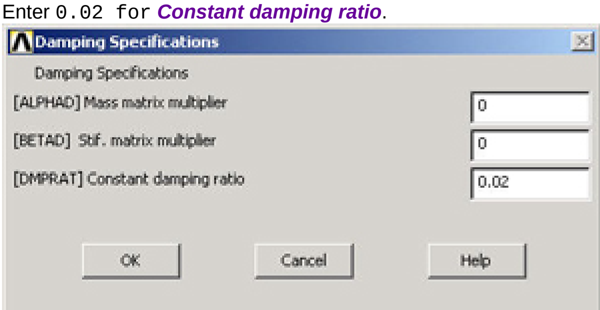



### Specify Damping Ratio

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



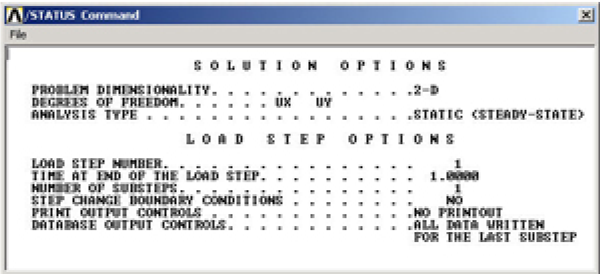
## Step 7: Solve!

### Enter Solution Module

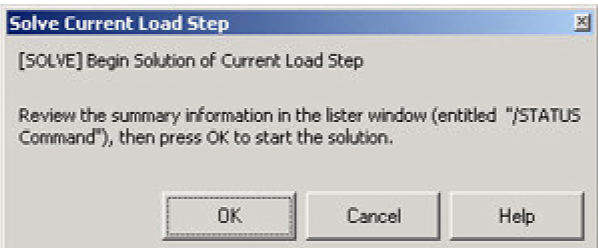
Select in *Main Menu:*

**Solution > Solve > Current LS**

Review the information in the /STAT *Command* window.




Close this window.




Click ***OK*** in *Solve Current Load Step* dialog box_._

ANSYS performs the solution and a yellow window should pop up saying "Solution is done!"

### Save your work

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

Go to Step 8: Postprocess the results

Go to all ANSYS Learning Modules



## Step 8: Postprocess the results

### Enter Postprocessing module to analyze solution

**Main Menu > General Postproc**

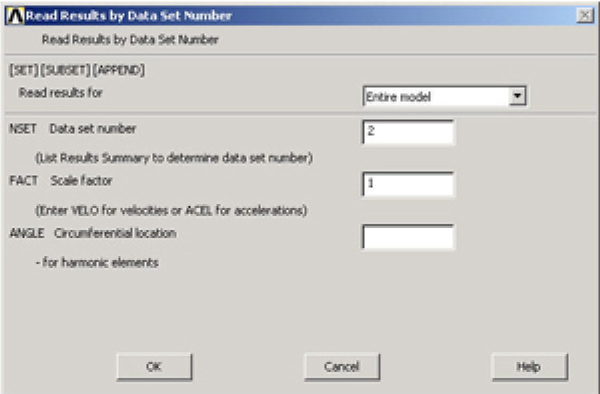
Select ***Results Summary***.

This shows you the cyclic frequencies of the ten modes. Compare with the values in the book.

View Mode Shape for Mode 2

**Read Results > By Set Numbers**

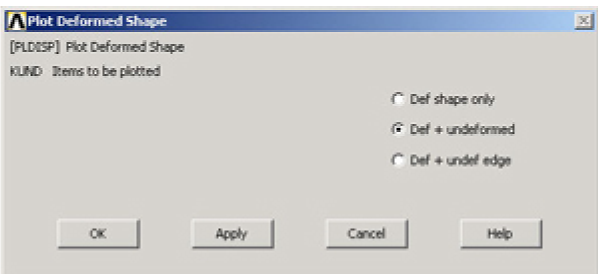
Enter 2 for ***NSET***.




Click ***OK***.

**Plot Results > Deformed Shape**

Select ***Def+undeformed***.




Click ***OK***.

This plots the mode shape for mode 2. Similarly, look at the other mode shape and compare them with figure 11.17-2 in the book.

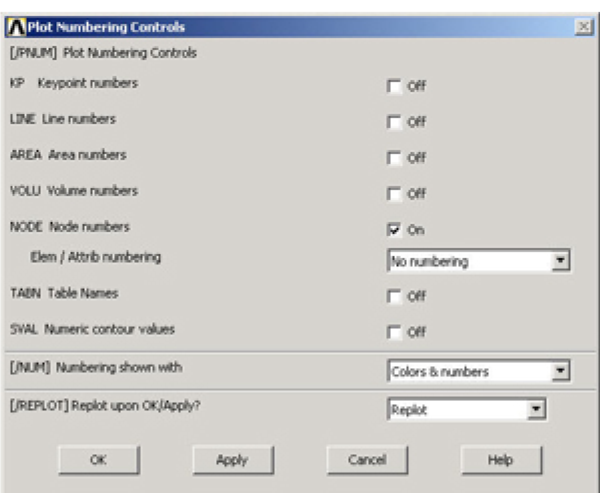
## Find Mode Numbers

Table 11.17-1 gives amplitude values for selected d.o.f. for three nodes.

To find the node numbers corresponding to the ones in the book, turn on node numbering.

**Utility Menu > PlotCtrls > Numbering**

Turn on ***Node Numbers***.




Click ***OK***.

If you need to refresh the screen: **Utility Menu > Plot > Multi-plots**

By comparing the node numbers, we find:

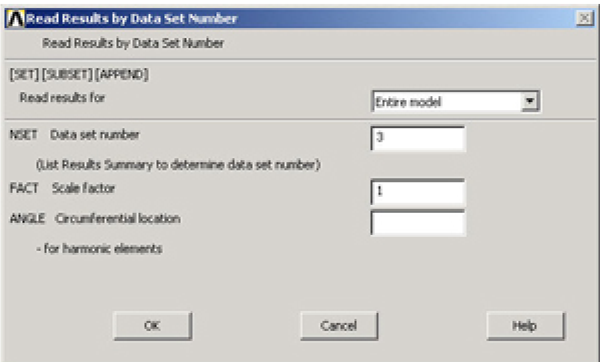
| Node Numbers | | Cook et al. | ANSYS |
|---|---|---|---|
| 16 | 17 | | |
| 41 | 42 | | |
| 51 | 32 | | |

### Determine the Displacement Amplitude

To determine the displacement amplitude at node 17 for mode 3,

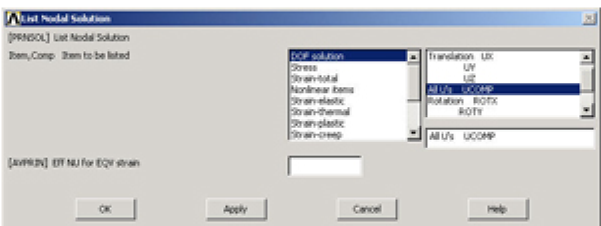
General **Post Proc > Read Results > By Set Number**

Enter 3 for ***NSET***.




General Post ***Proc > List Results > Nodal Solution***

Select ***UCOMP***.




From the list, the displacement amplitude, denoted as ***USUM***, is 23.9e-3. The corresponding value in table 11.17-1 is 23.8e-3. Similarly, you can determine the other entries in the table. Note that the rotational d.o.f. to use for the second row in the table is ***ROTZ***_._

### Save your work

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

Go to Step 9: Validate the results

Go to all ANSYS Learning Modules



## Step 9: Validate the results

Not available yet.

Back to Problem Specification

Go to all ANSYS Learning Modules

Go to all ANSYS Learning Modules