

ANSYS - Vibration Analysis of a Frame

- Problem Specification
- Step 1: Start-up and preliminary set-up
 - Create a folder
 - Start ANSYS
 - Specify directory and job name
 - Set Preferences
 - Enter Parameters
- Step 2: Specify element type and constants
 - Specify Element Type
 - Specify the Constants
- Step 3: Specify material properties
 - Enter the Define Material Model Behavior menu
 - Specify Material properties
 - Save your work
- Step 4: Specify geometry
 - Create Keypoints
 - Create the Lines AB and BC
 - Save your work
- Step 5: Mesh geometry
 - Generate the Mesh
 - Define Number of Elements for Each Line
 - Creating the Mesh
 - Save your work
- Step 6: Specify boundary conditions
 - Set Options
 - Apply Displacement Constraints
 - Specify Damping Ratio
 - Save your work
- Step 7: Solve!
 - Enter Solution Module
 - Save your work
- Step 8: Postprocess the results
 - Enter Postprocessing module to analyze solution
 - Find Mode Numbers
 - Determine the Displacement Amplitude
 - Save your work
- Step 9: Validate the results

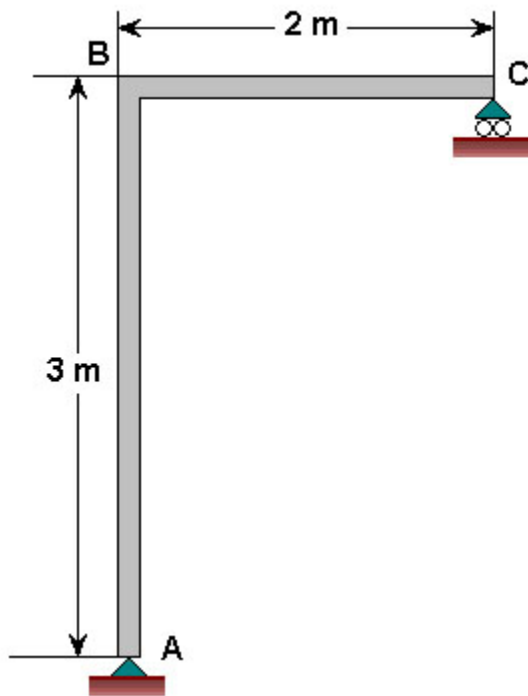
Author: Rajesh Bhaskaran, Cornell University

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

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.](#)



$$E = 200 \times 10^9 \text{ Pa}$$

$$\nu = 0.29$$

$$\rho = 7860 \text{ kg/m}^3$$

$$I = \frac{1 \times 10^{-4}}{12}$$

Go to [Step 1: Start-up and preliminary set-up](#)

[Go to all ANSYS Learning Modules](#)

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 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

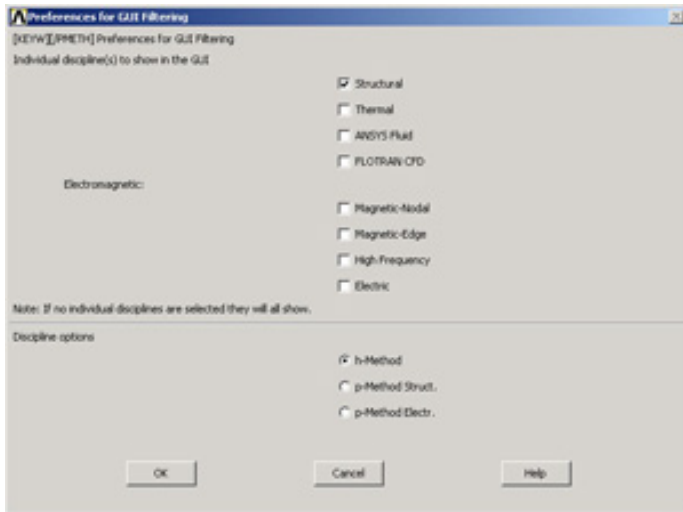
Specify directory and job name

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 ν , I , ρ , 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](#)

- 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

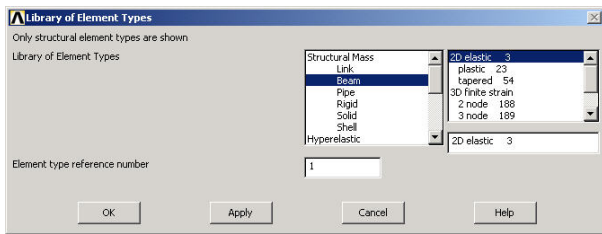
Step 2: Specify element type and constants

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*:*

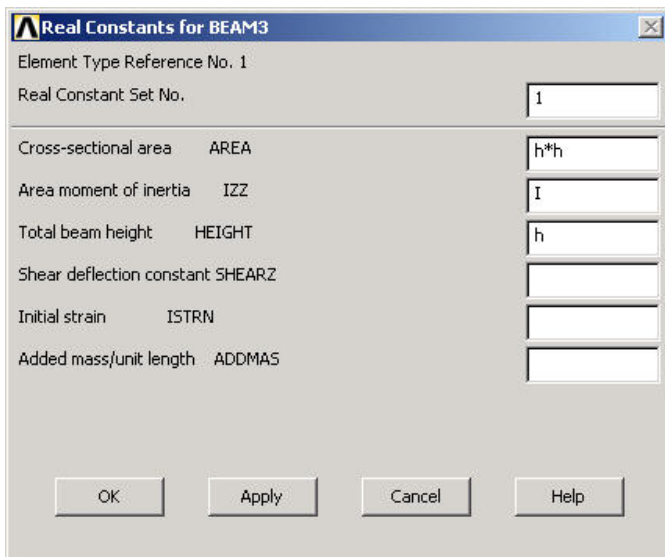
Real Constants > Add/Edit/Delete > Add...

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](#)

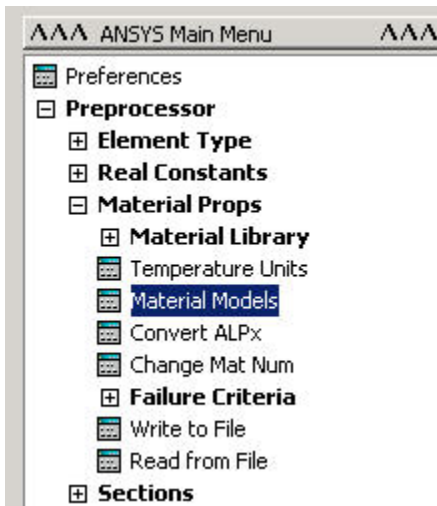
Problem Specification

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

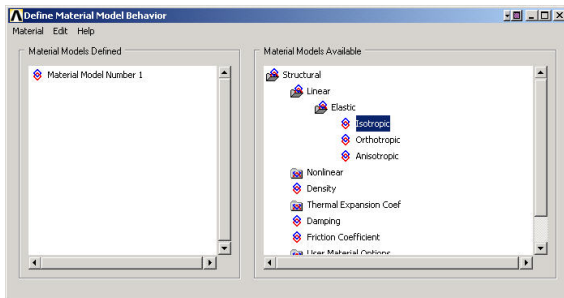
Step 3: Specify material properties

Enter the Define Material Model Behavior menu

Select Main Menu > Preprocessor > Material Props > Material Models

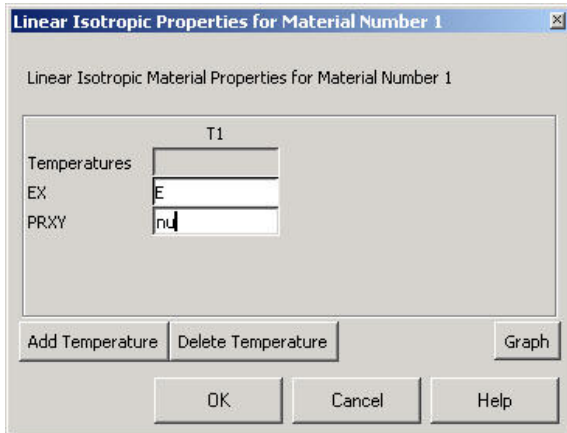


In the *Define Material Model Behavior* menu, double-click on *Structural, Linear, Elastic, and Isotropic*.



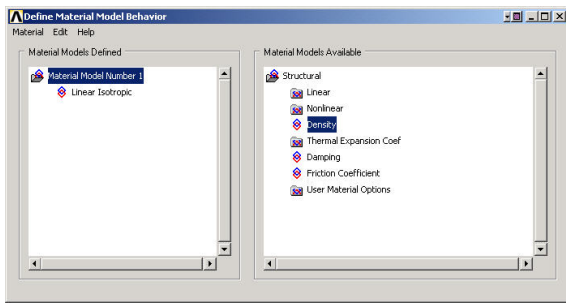
Specify Material properties

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

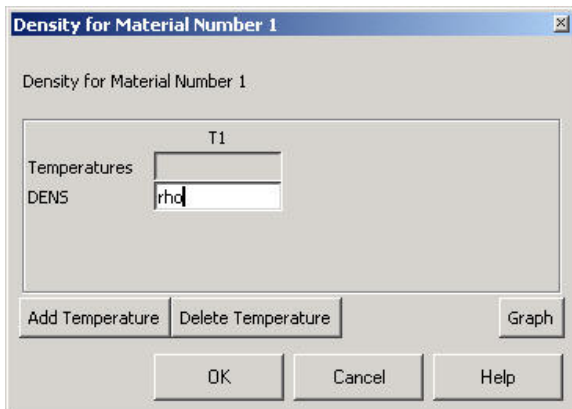


Click *OK*.

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](#)

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:

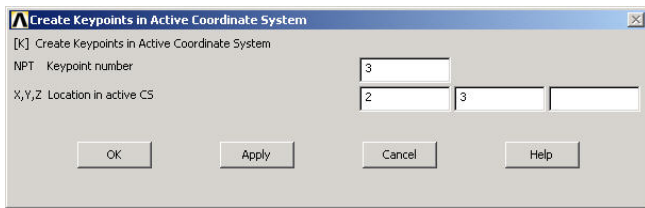
Modeling > Create > Keypoints > In Active CS

Enter:

Keypoint 1: X=0, Y=0

Keypoint 2: X=0, Y=3

Keypoint 3: X=2, Y=3



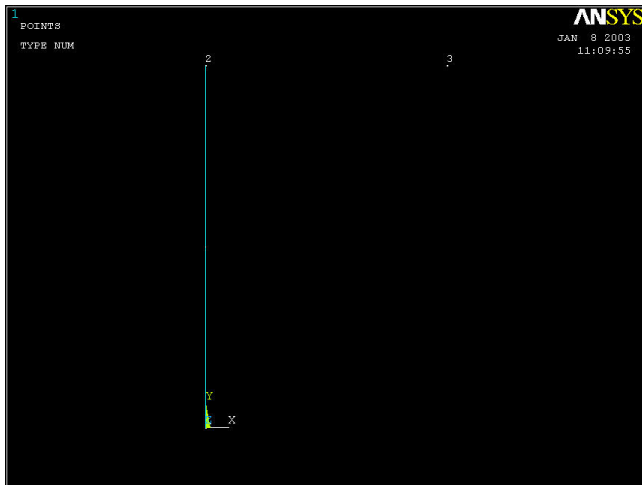
Click **OK**.

Create the Lines AB and BC

Select in *Preprocessor* menu:

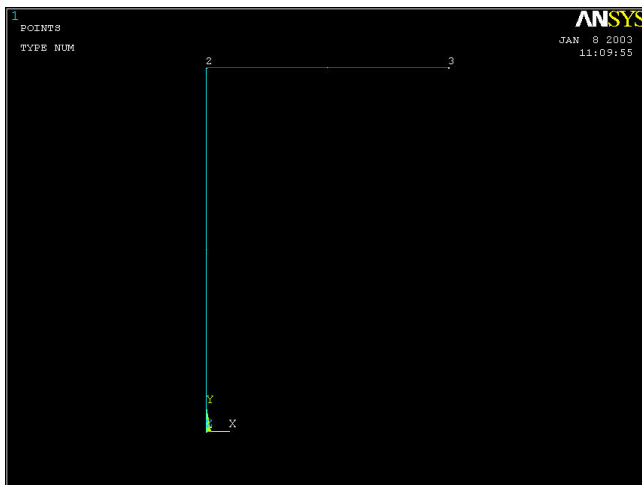
Modeling > Create> Lines > Lines > In Active Coord

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](#)

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 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.

MeshTool

Element Attributes:

Global

Set

☐ Smart Size

Fine6Coarse

Size Controls:

Global

Set

Clear

Areas

Set

Clear

Lines

Set

Clear

Copy

Flip

Layer

Set

Clear

Keypts

Set

Clear

Mesh:

Lines

Shape:

☒ Radio1

☐ Hex/wedge

☒ Free

☐ Mapped

☐ Sweep

3 or 4 sided

Mesh

Clear

Refine at:

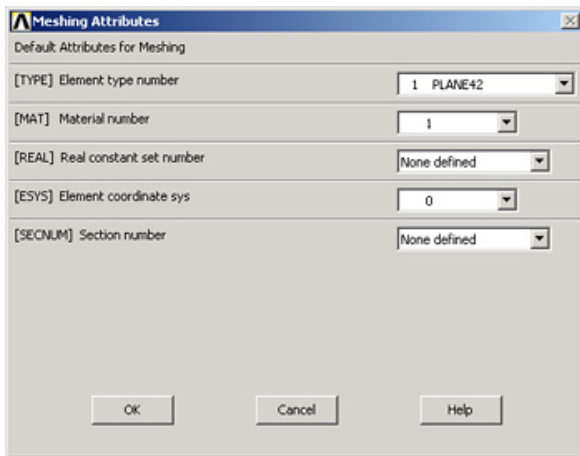
Elements

Refine

Close

Help

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.

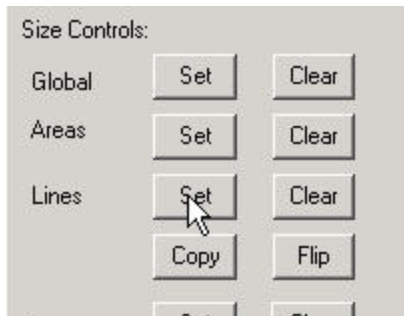


Close this menu by clicking **OK**.

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**.

Element Sizes on Picked Lines

[LESIZE] Element sizes on picked lines

SIZE Element edge length

NDIV No. of element divisions

(NDIV is used only if SIZE is blank or zero)

KYNDIV SIZE,NDIV can be changed ☒ Yes

SPACE Spacing ratio

ANGSIZ Division arc (degrees)

(use ANGSIZ only if number of divisions (NDIV) and element edge length (SIZE) are blank or zero)

Clear attached areas and volumes ☐ No

OK Apply Cancel Help

Click **Apply**.

Select line BC.

Min, Max, Inc

OK Apply

Reset Cancel

Pick All Help

Click **OK**.

Enter 20 for **NDIV**.

Element Sizes on Picked Lines

[LESIZE] Element sizes on picked lines

SIZE Element edge length

NDIV No. of element divisions

(NDIV is used only if SIZE is blank or zero)

KYNDIV SIZE,NDIV can be changed ☒ Yes

SPACE Spacing ratio

ANGSIZ Division arc (degrees)

(use ANGSIZ only if number of divisions (NDIV) and element edge length (SIZE) are blank or zero)

Clear attached areas and volumes ☐ No

OK Apply Cancel Help

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](#)

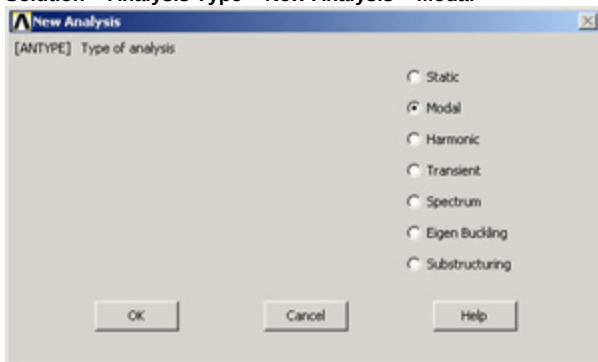
- Problem Specification
1. Start-up and preliminary set-up
 2. Specify element type and constants
 3. Specify material properties
 4. Specify geometry
 5. Mesh geometry
 - 6. Specify boundary conditions**
 7. Solve!
 8. Postprocess the results
 9. Validate the results

Step 6: Specify boundary conditions

Set Options

Select in *Main Menu*:

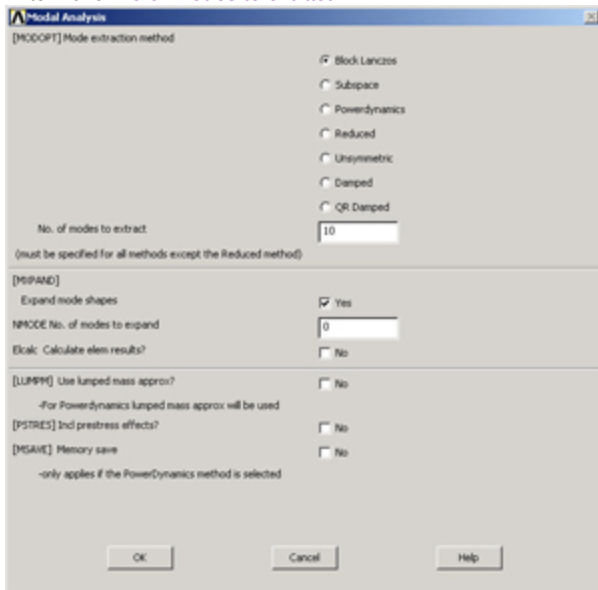
Solution > Analysis Type > New Analysis > Modal



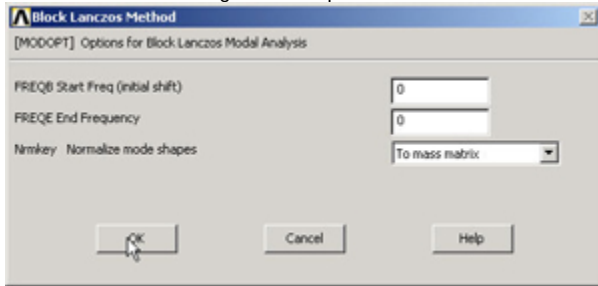
Then select in *Main Menu*:

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*.

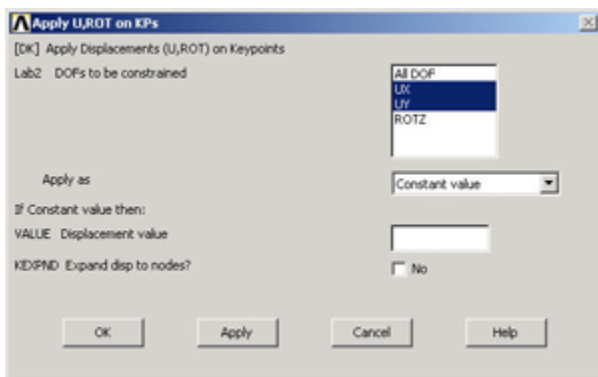


Apply Displacement Constraints

Select in *Preprocessor*:

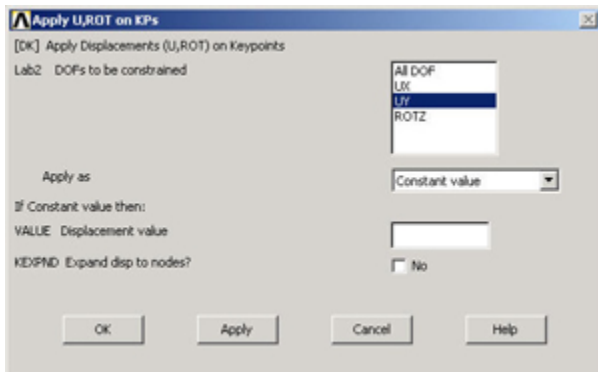
Loads > Define Loads > Apply > Structural > Displacement > On Keypoints

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

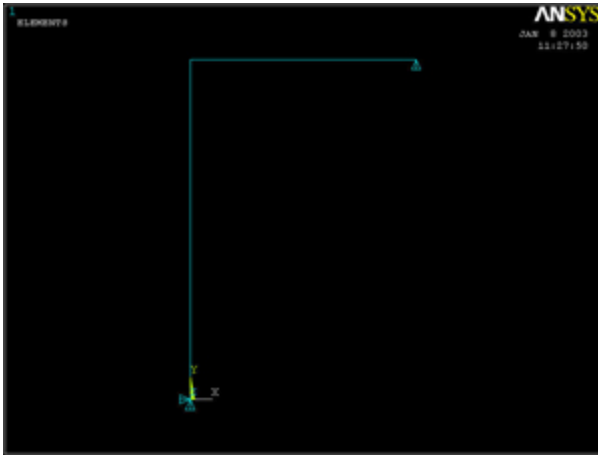


Click **OK**.

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



Click **OK**.

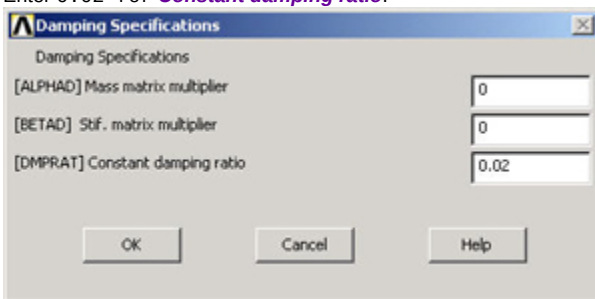


Specify Damping Ratio

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](#)

- 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

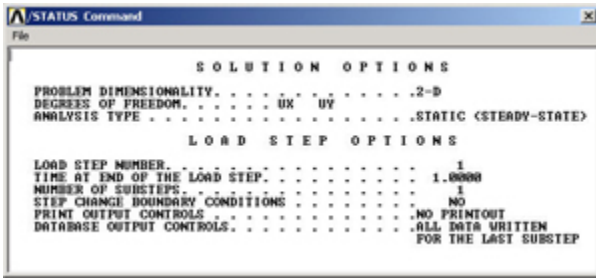
Step 7: Solve!

Enter Solution Module

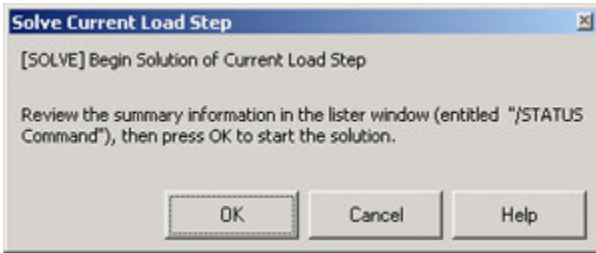
Select in *Main Menu*:

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](#)

- 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 8: Postprocess the results

Enter Postprocessing module to analyze solution

Main Menu > General Postproc

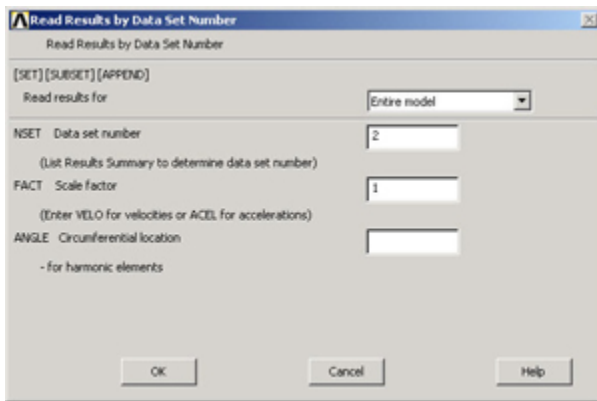
Select **Results Summary**.

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

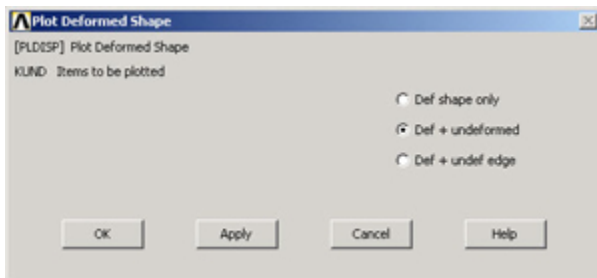
Enter 2 for **NSSET**.



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.

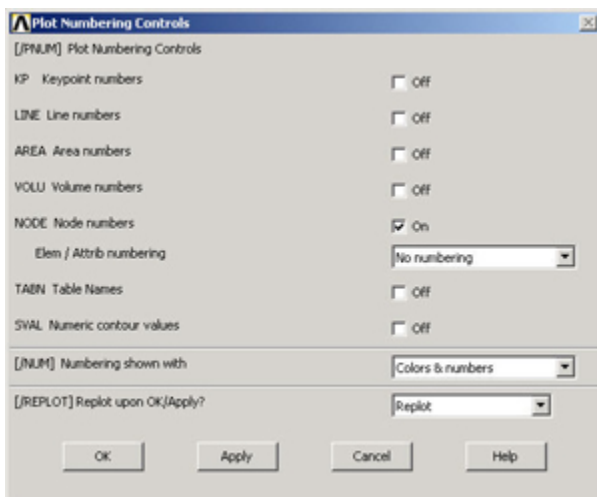
Find Mode Numbers

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:

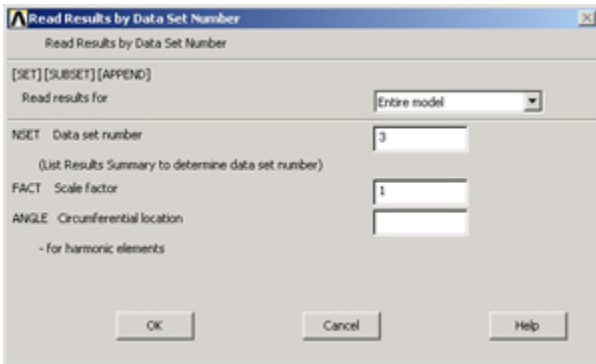
Node Numbers		Cook et al.	ANSYS
16	17		
41	42		
51	32		

Determine the Displacement Amplitude

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

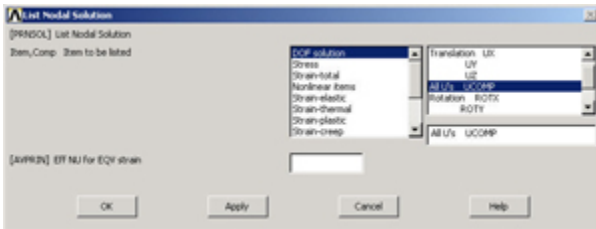
General **Post Proc** > **Read Results** > **By Set Number**

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](#)

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 9: Validate the results

Not available yet.

Back to [Problem Specification](#)

Go to [all ANSYS Learning Modules](#)

Go to [all ANSYS Learning Modules](#)