# **ANSYS - Vibration Analysis of a Frame - Step 8**

- **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 NSET.

[SET] [SUBSET] [APPEND]		
Read results for	Entire model	-
NSET Data set number	2	
(List Results Summary to determine data set number)		
FACT Scale factor	1	
(Enter VELO for velocities or ACEL for accelerations)		
ANGLE Circumferential location		
- for harmonic elements		

Click OK.

### Plot Results > Deformed Shape

#### Select Def+undeformed.

APlot Deformed Shape		X
[PLDISP] Plot Deformed Shape		
KUND Items to be plotted		
	C Def	shape only
	(* Def	+ undeformed
	C Def	+ undef edge
ОК Арріу	Cancel	Help

## 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.

Plot Numbering Controls	×
[/PNUM] Plot Numbering Controls	
KP Keypoint numbers	C Off
LINE Line numbers	IT Off
AREA Area numbers	I ⊂ off
VOLU Volume numbers	I⊏ off
NODE Node numbers	🖓 On
Elem / Attrib numbering	No numbering 💌
TABN Table Names	I ⊂ off
SVAL Numeric contour values	r off
[INUM] Numbering shown with	Colors & numbers
[/REPLOT] Replot upon OK/Apply?	Replot
ОК Аррју	Cancel Help

## 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.	ANSY S
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.

Read Results by Data Set Number		
[SET] [SUBSET] [APPEND]		
Read results for	Entire model	*
NSET Data set number	3	
(List Results Summary to determine data	set number)	
FACT Scale factor	1	
AVGLE Circumferential location		
- for harmonic elements		
OK	Cancel	Help

General Post Proc > List Results > Nodal Solution

## Select UCOMP.

(PRNSOL) List Nodel Solution					
Zen,Corp Zen to be lated		Boul asilition Stress Strain-total Nonlinear Items Strain-themail Strain-themail Strain-creep	-	Translation UX UV U2 MU/n U02049 Rotation ROTX BOTY All U5 U2204P	•
[AvPR2N] ET NU for EQF strain					
OK	Apply	Cancel		Help	

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