ANSYS®

Galloping Gertie

Introductory Workshop
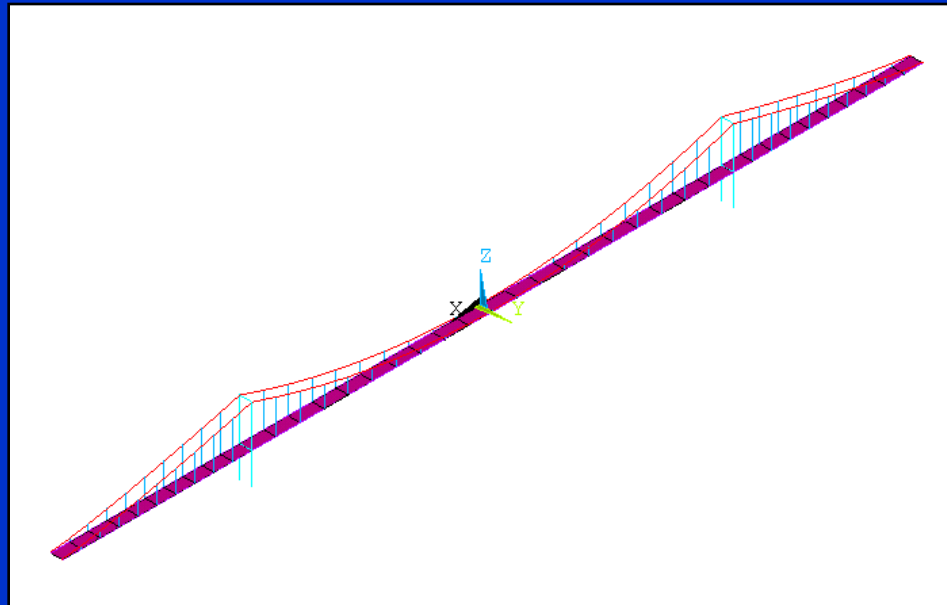
Instructions

1. **Enter ANSYS in the working directory specified by your instructor.**

2. **Open the "gallop.db" database file:**

   Utility Menu > Open ANSYS File   choose "gallop.db"
   - **This will create the model and perform a static analysis to prestress the bridge.**
   - **The next step is to do a modal analysis.**

3. **Enter Solution and change analysis type to Modal:**

    Solution > Analysis Type > New Analysis…   **choose Modal.**

4. **Set the following analysis options.**

    Solution > Analysis Type > Analysis Options...

    accept the default (Block Lanczos)

    10 modes to extract

    10 modes to expand

    Calculate element stresses

    Include prestress effects… press OK
    Accept defaults on the next dialog (Options for Block Lanczos Modal Analysis)
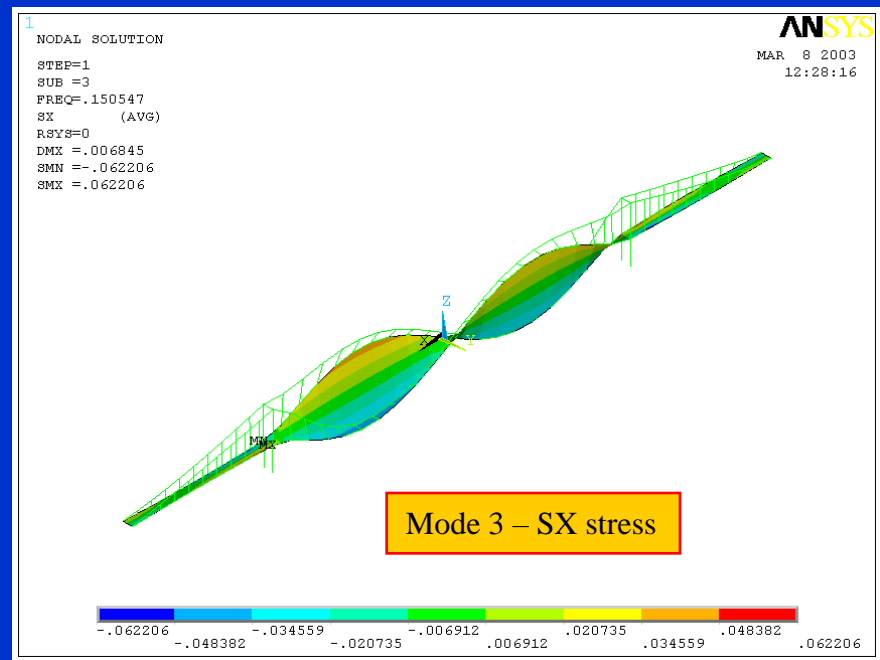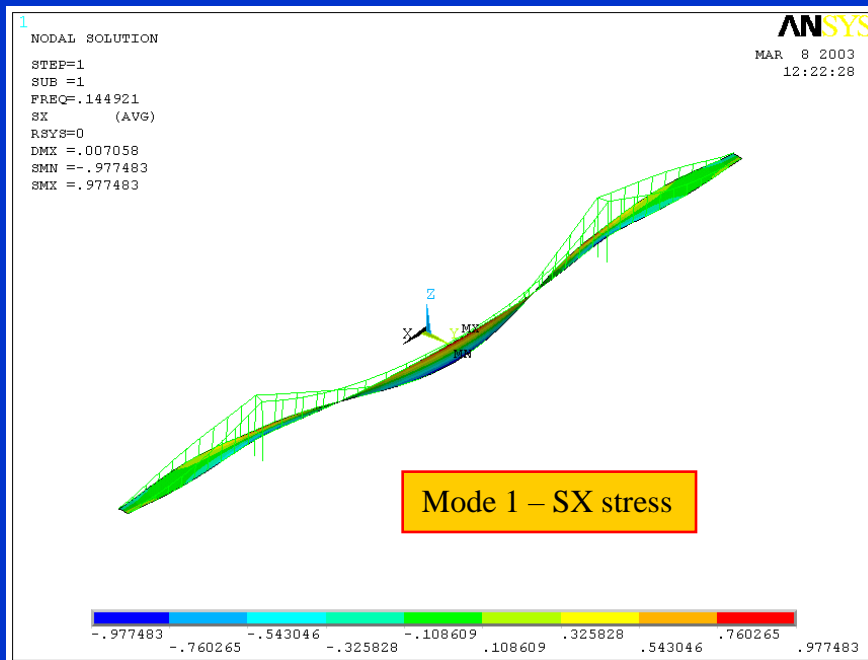
5. **Solve.**

    Solution > Solve > Current LS

6. Plot the first few mode shapes.

General Postproc > Read Results > By Pick …

General Postproc > Plot Results > Contour Plot > Nodal Solu ...

Mode 1 – SX stress

Mode 3 – SX stress

List the natural frequencies

- General Postproc > Results Summary
- **Or issue** SET,LIST
- **Note that each mode is stored in a separate *substep*.**


```
 SET,LIST Command
 File

***** INDEX OF DATA SETS ON RESULTS FILE *****

 SET   TIME/FREQ    LOAD STEP    SUBSTEP    CUMULATIVE
   1  0.14492           1           1           1
   2  0.14948           1           2           2
   3  0.15055           1           3           3
   4  0.16346           1           4           4
   5  0.17049           1           5           5
   6  0.21640           1           6           6
   7  0.21677           1           7           7
   8  0.21687           1           8           8
   9  0.21739           1           9           9
  10  0.23903           1          10          10
```

DYNAMICS 7.0

7. **Enter Solution and choose harmonic analysis.**

   Solution > Analysis Type > New Analysis…

8. **Set the following analysis options.**

   Solution > Analysis Type > Analysis Options...

   Select the Mode superposition solution method

   Defaults for all others (including subsequent dialog box)

9. **Set frequency and substep options:**

   Solution > Load Step Opts > Time/Frequenc > Freq and Substps...

   Harmonic frequency range = 0 to 0.4

   Number of substeps = 40

   Stepped boundary conditions

10. **Set constant damping ratio = 0.01.**

    Solution > Load Step Opts > Time/Frequenc > Damping…

... Galloping Gertie

11. Apply a load vector for mode superposition with a scale factor of 100.

Solution > Define Loads > Apply > Load Vector > For Mode Super…
**(close the warning message window)**

12. Solve: Solution > Solve > Current LS

13. Save the ANSYS database for the Variable Viewer in Step 14.

Utility Menu: File > Save as Jobname.db …

14. Enter POST26 (TimeHist Postproc). The Variable Viewer will start automatically. Specify the results file name, i.e. gallop.rfrq, by clicking on File > Open Results)

Select "gallop.rfrq" as the results file, then click [Open]
Select "gallop.db" as the ANSYS database, then click [Open]
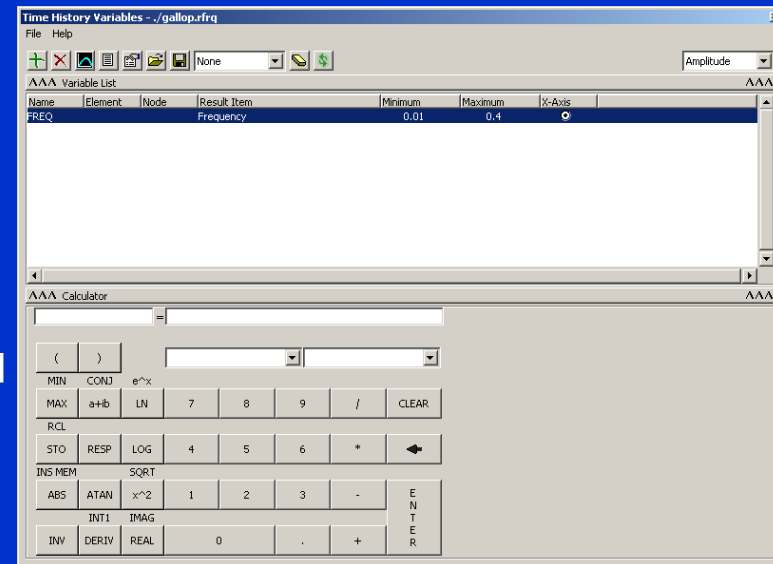
15. Create a scalar parameter to represent the center node: At command line type in ncen = node(0,0,0) .

16. Define a variable (a vector) using the Variable Viewer that will contain the UZ displacements of the center node:

a. **Click on the "Add Data" button**

b. **Double click on "Nodal Solution" and "DOF Solution", select "Z-Component of displacement" and enter "uz_mid" for the Variable Name, and then click [OK]**

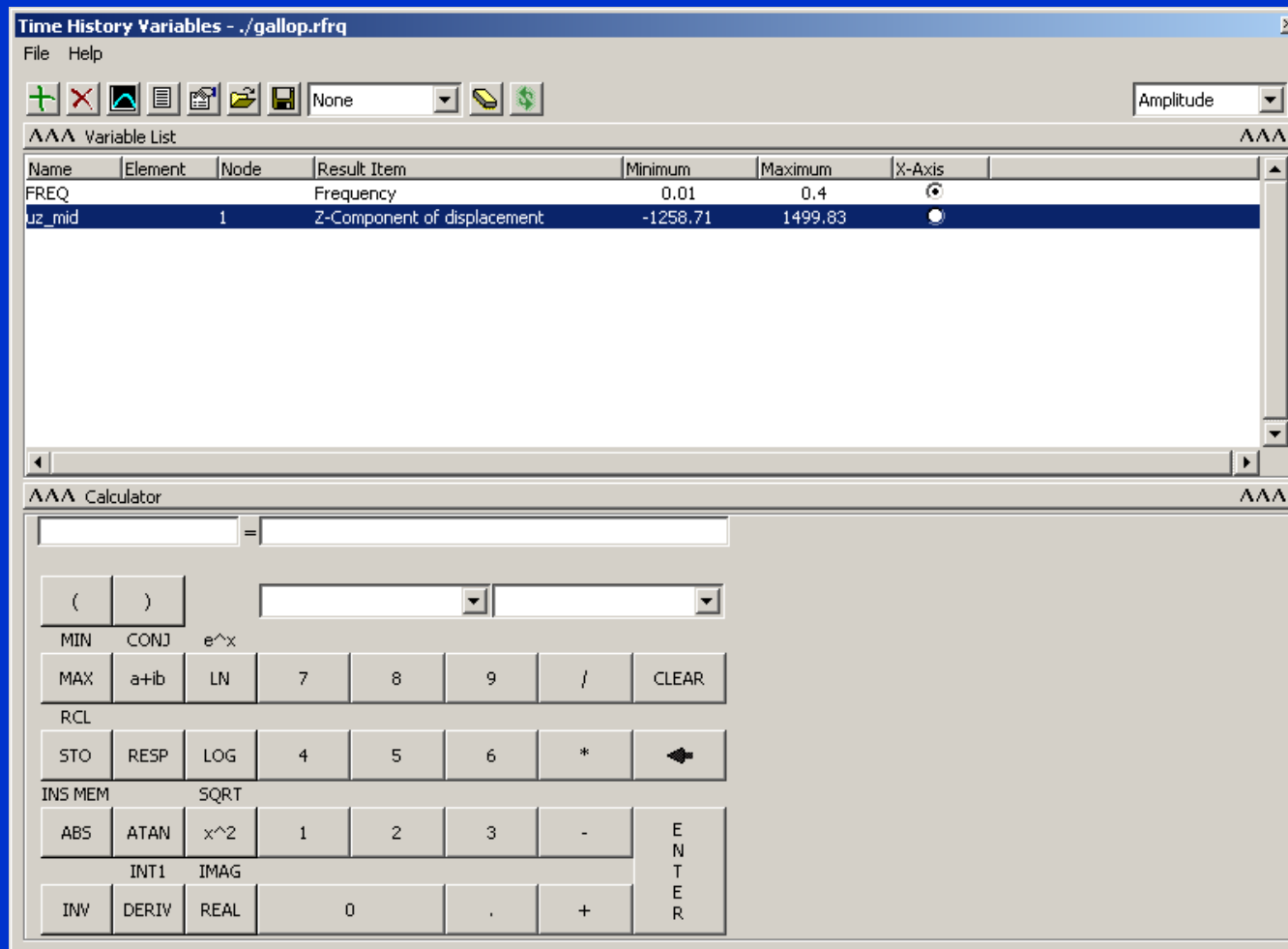c. **Enter "ncen" followed by [Enter] in the ANSYS Picker Menu, then [OK]**
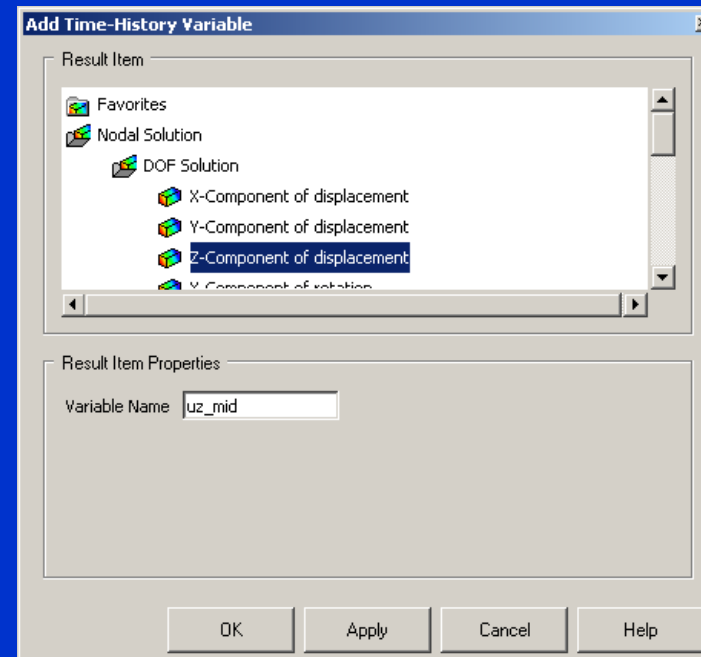
16. (cont'd).

The Variable Viewer should appear as follows:

17. Graph the UZ-displacement vs frequency:

1. Select the line labeled "uz_mid" and then click on the "Graph Data" button
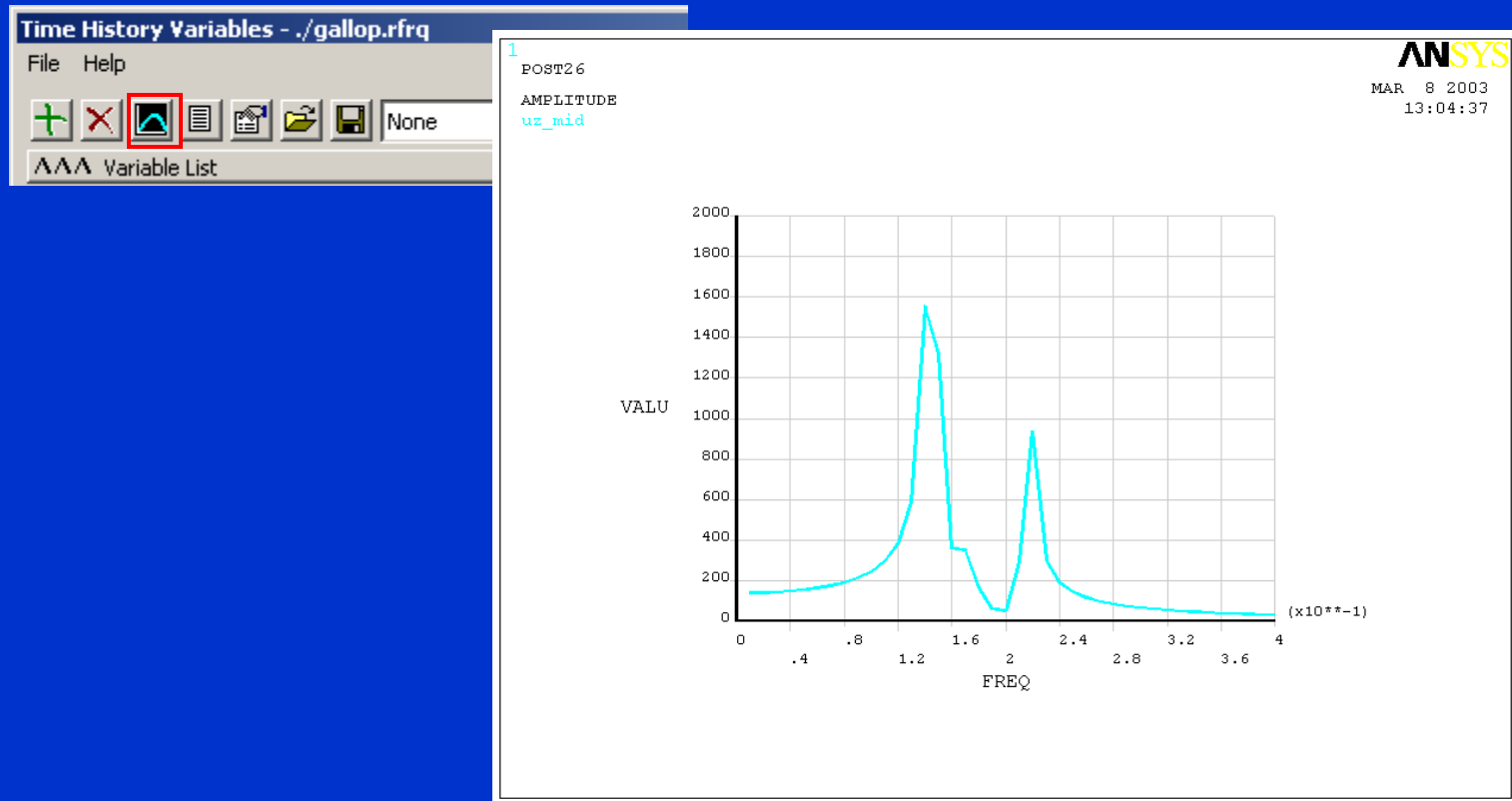
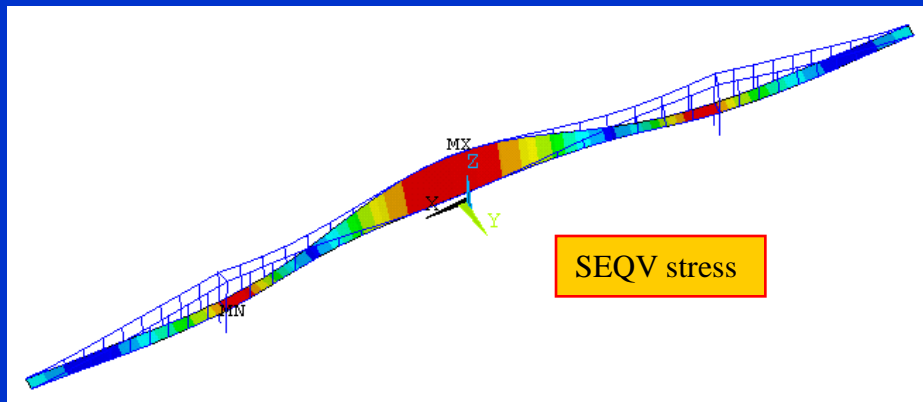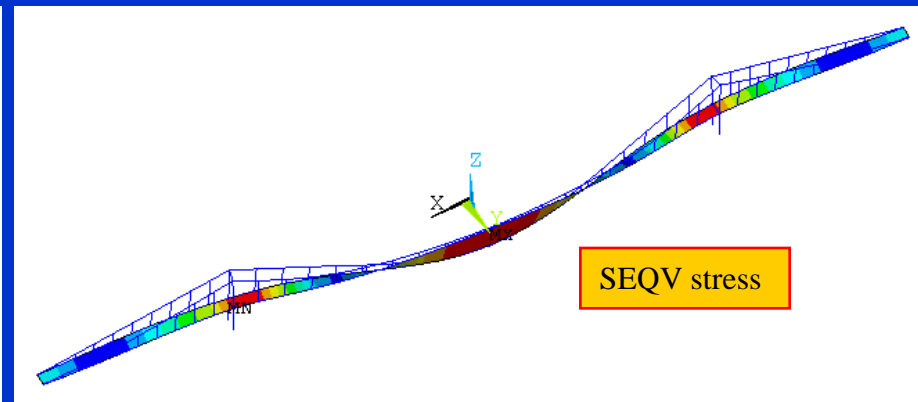**18. Close the Variable Viewer and then Exit ANSYS or go to step 19 if time permits.**

Optional: Continue with the following steps to review the deformed shape and stresses at 0.07 Hz frequency.

19. Read Input from… gallop_more.inp.

20. Enter POST1, read results for load step 1 substep 7, and plot the deformed shape and stress contours. Repeat for the imaginary part as well.

21. Exit ANSYS.

Real Part

Imaginary Part