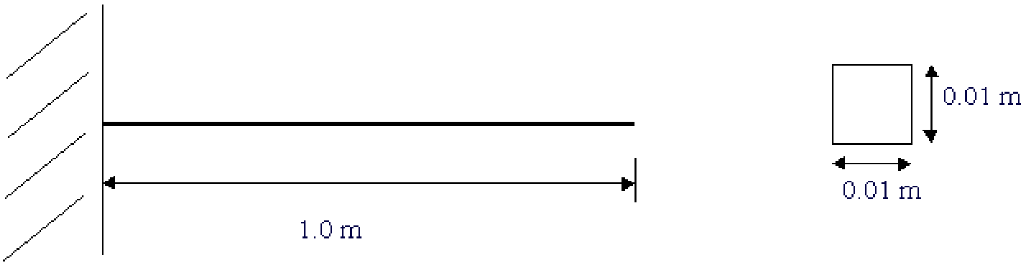
Harmonic Analysis of a Cantilever Beam

Introduction

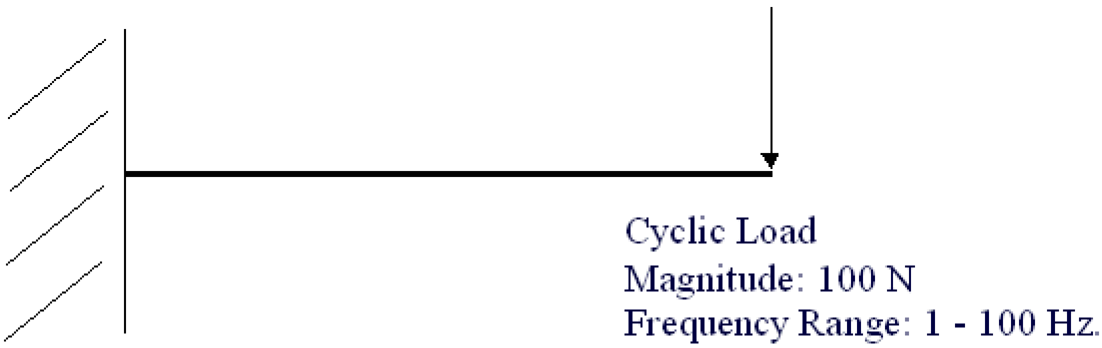
This tutorial was created using ANSYS 7.0 The purpose of this tutorial is to explain the steps required to perform Harmonic analysis the cantilever beam shown below.



Modulus of Elasticity (E) = 206800(10^6) N/m^2

Density = 7830 kg/m^3

We will now conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1 - 100 Hz. The figure below depicts the beam with the application of the load.



ANSYS provides 3 methods for conducting a harmonic analysis. These 3 methods are the **Full** , **Reduced** and **Modal Superposition** methods.
This example demonstrates the **Full** method because it is simple and easy to use as compared to the other two methods. However, this method makes use of the full stiffness and mass matrices and thus is the slower and costlier option.

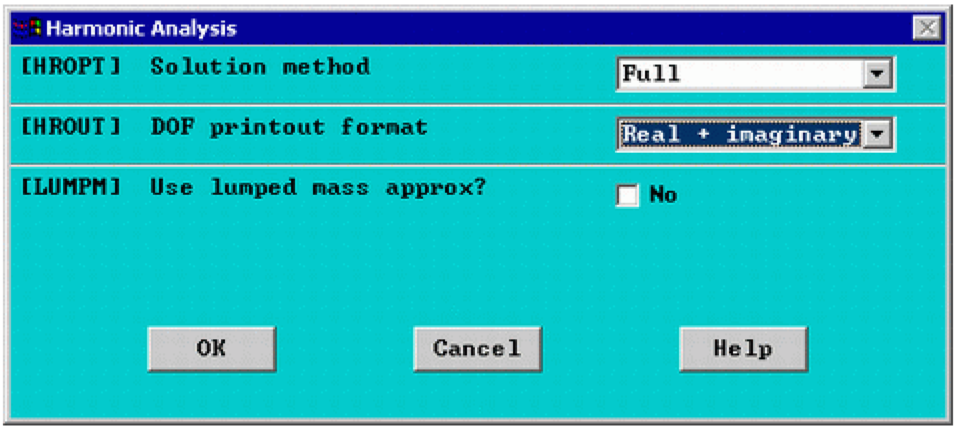
Preprocessing: Defining the Problem

The simple cantilever beam is used in all of the Dynamic Analysis Tutorials. If you haven't created the model in ANSYS, please use the links below. Both the command line codes and the GUI commands are shown in the respective links.

Solution: Assigning Loads and Solving

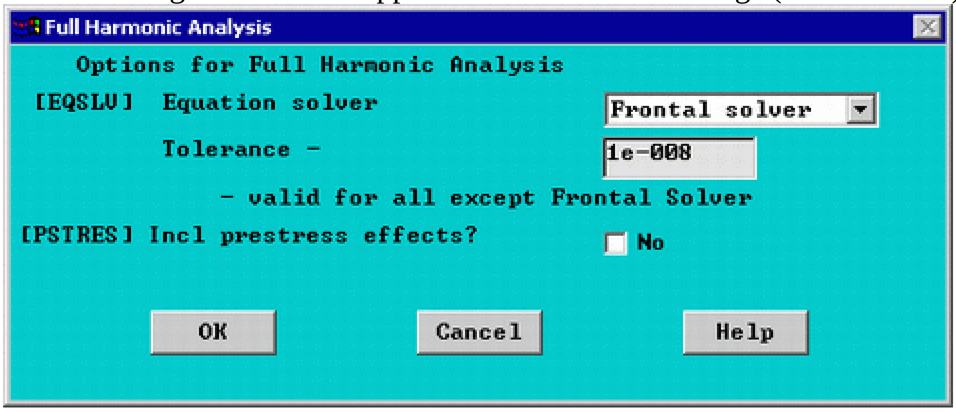
1. **Define Analysis Type (Harmonic)**
   Solution > Analysis Type > New Analysis > Harmonic
   `ANTYPE,3`
2. **Set options for analysis type:**
   - Select: **Solution > Analysis Type > Analysis Options..**

   The following window will appear



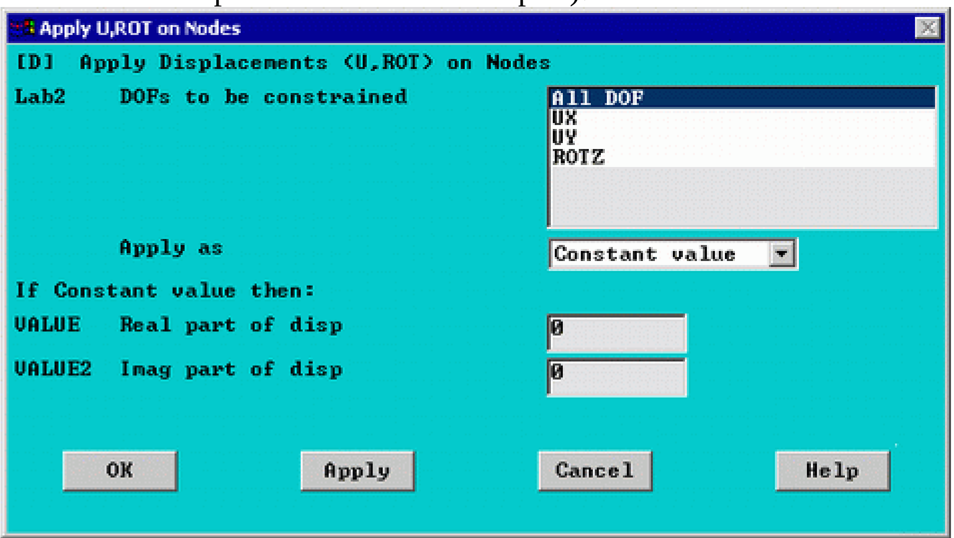
- o As shown, select the **Full** Solution method, the **Real + imaginary** DOF printout format and do not use lumped mass approx.
- o Click 'OK'

The following window will appear. Use the default settings (shown below).

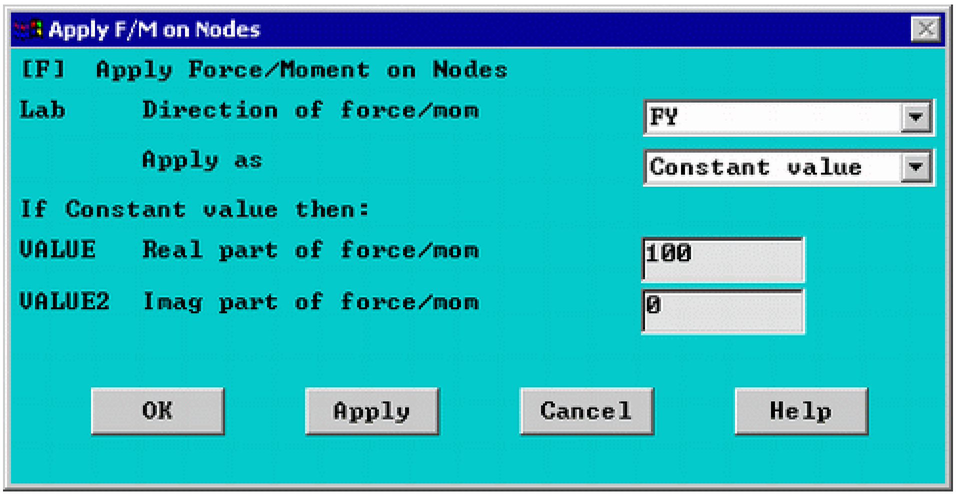


3. **Apply Constraints**
   - o Select **Solution > Define Loads > Apply > Structural > Displacement > On Nodes**

   The following window will appear once you select the node at x=0 (Note small changes in the window compared to the static examples):

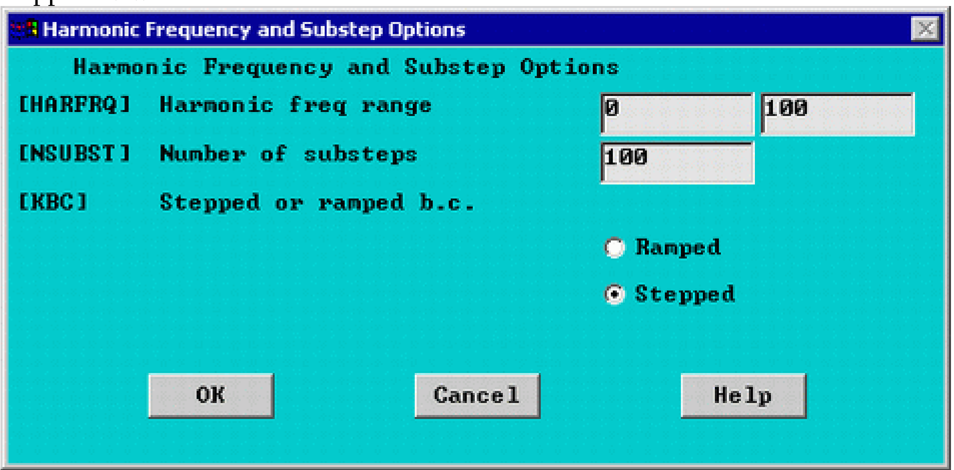


   - o Constrain all DOF as shown in the above window
4. **Apply Loads:**
   - o Select **Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes**
   - o Select the node at x=1 (far right)
   - o The following window will appear. Fill it in as shown to apply a load with a real value of 100 and an imaginary value of 0 in the positive 'y' direction



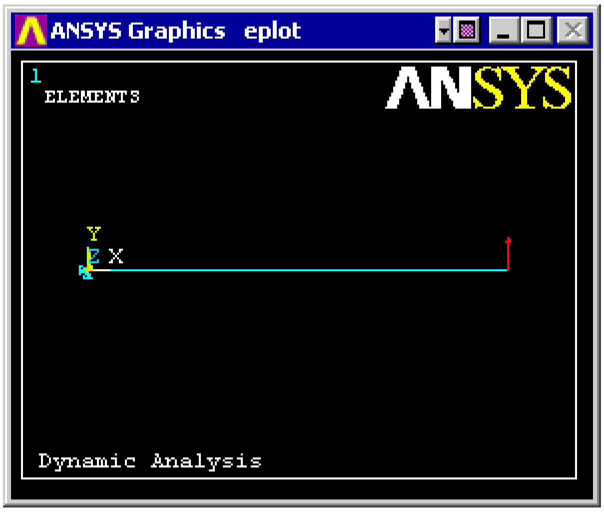
Note: By specifying a real and imaginary value of the load we are providing information on magnitude and phase of the load. In this case the magnitude of the load is 100 N and its phase is 0. Phase information is important when you have two or more cyclic loads being applied to the structure as these loads could be in or out of phase. For harmonic analysis, all loads applied to a structure must have the **SAME FREQUENCY**.

5. **Set the frequency range**
   - o Select **Solution > Load Step Opts > Time/Frequency > Freq and Substps...**
   - o As shown in the window below, specify a frequency range of 0 - 100Hz, 100 substeps and stepped b.c..



By doing this we will be subjecting the beam to loads at 1 Hz, 2 Hz, 3 Hz, ... 100 Hz. We will specify a stepped boundary condition (KBC) as this will ensure that the same amplitude (100 N) will be applyed for each of the frequencies. The ramped option, on the other hand, would ramp up the amplitude where at 1 Hz the amplitude would be 1 N and at 100 Hz the amplitude would be 100 N.

You should now have the following in the ANSYS Graphics window



6. **Solve the System**
Solution > Solve > Current LS
SOLVE

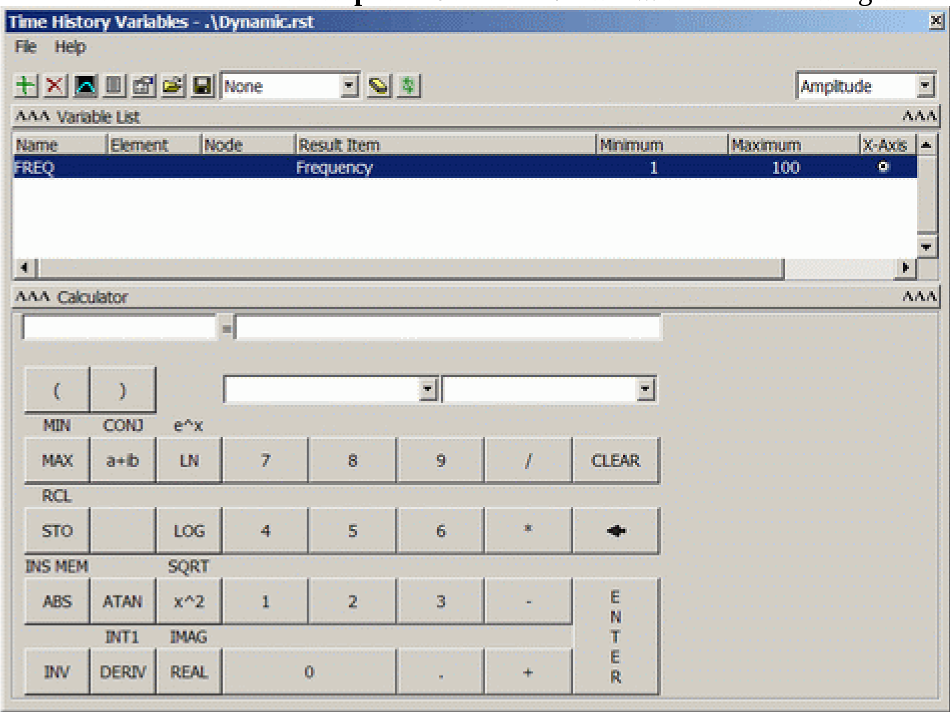
Postprocessing: Viewing the Results

We want to observe the response at x=1 (where the load was applyed) as a function of frequency. We cannot do this with **General PostProcessing (POST1)**, rather we must use **TimeHist PostProcessing (POST26)**. POST26 is used to observe certain variables as a function of either time or frequency.

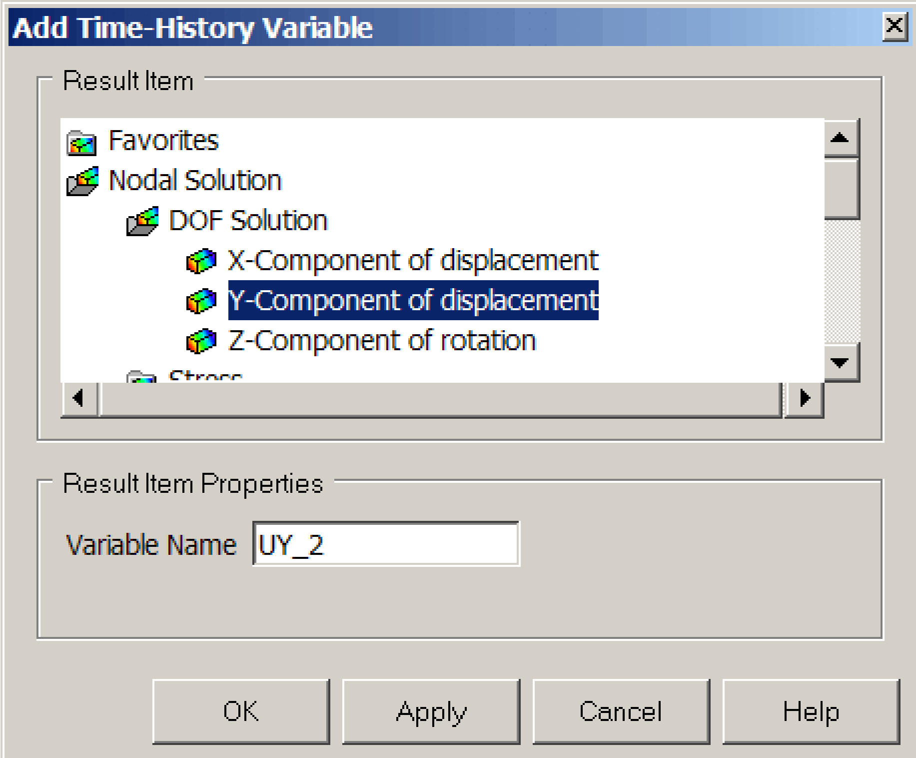
1. **Open the TimeHist Processing (POST26) Menu**
Select **TimeHist Postpro** from the ANSYS Main Menu.

2. **Define Variables**
In here we have to define variables that we want to see plotted. By default, **Variable 1** is assigned either **Time** or **Frequency**. In our case it is assigned **Frequency**. We want to see the displacement UY at the node at x=1, which is node #2. (To get a list of nodes and their attributes, select **Utility Menu > List > nodes**).

   o Select **TimeHist Postpro > Variable Viewer...** and the following window should pop up.

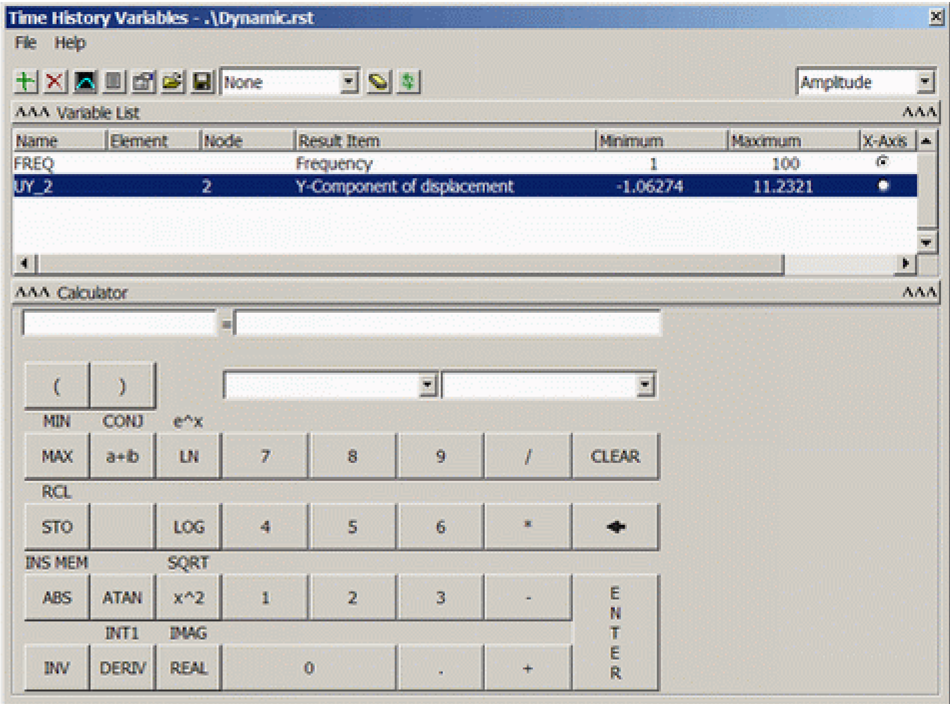


   o Select Add (the green '+' sign in the upper left corner) from this window and the following window should appear



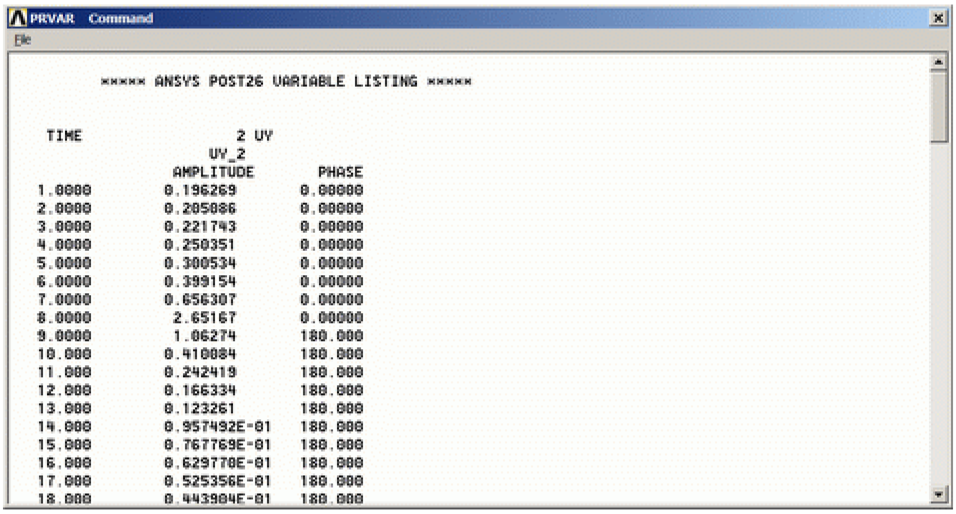
o We are interested in the **Nodal Solution > DOF Solution > Y-Component of displacement**. Click OK.

o Graphically select node 2 when prompted and click OK. The 'Time History Variables' window should now look as follows



3. **List Stored Variables**

o In the 'Time History Variables' window click the 'List' button, 3 buttons to the left of 'Add' The following window will appear listing the data:



PRVAR Command

File

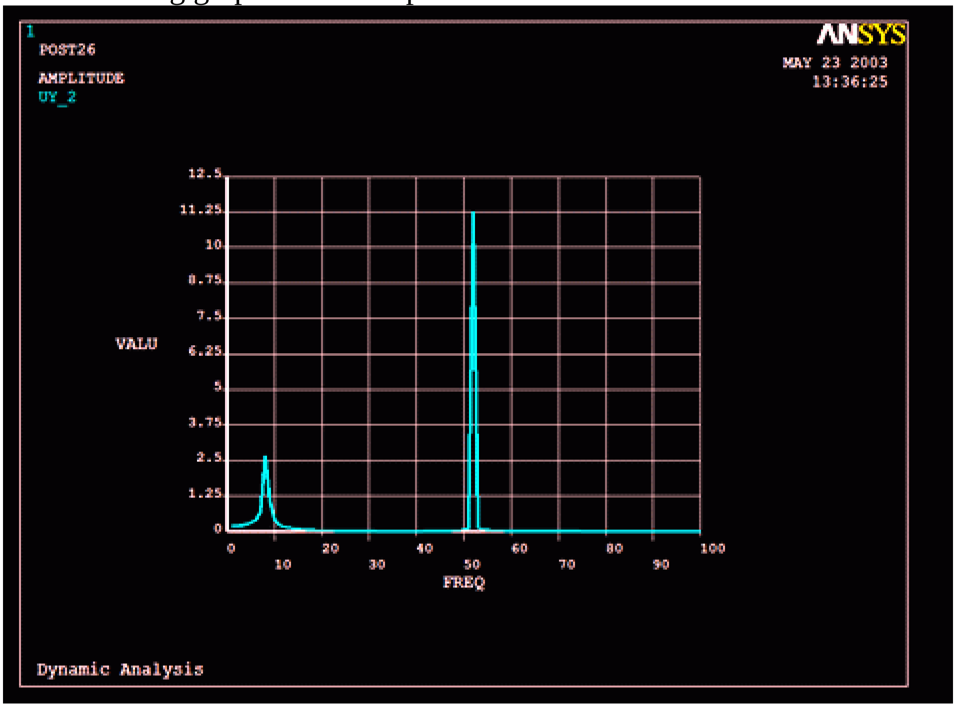
***** ANSYS POST26 VARIABLE LISTING *****

TIME	2 UY UY_2 AMPLITUDE	PHASE
1.0000	0.196269	0.00000
2.0000	0.205086	0.00000
3.0000	0.221743	0.00000
4.0000	0.250351	0.00000
5.0000	0.300534	0.00000
6.0000	0.399154	0.00000
7.0000	0.656307	0.00000
8.0000	2.65167	0.00000
9.0000	1.06274	180.000
10.000	0.410084	180.000
11.000	0.242419	180.000
12.000	0.166334	180.000
13.000	0.123261	180.000
14.000	0.957492E-01	180.000
15.000	0.767769E-01	180.000
16.000	0.629770E-01	180.000
17.000	0.525356E-01	180.000
18.000	0.443984E-01	180.000

4. **Plot UY vs. frequency**

o In the 'Time History Variables' window click the 'Plot' button, 2 buttons to the left of 'Add'

The following graph should be plotted in the main ANSYS window.

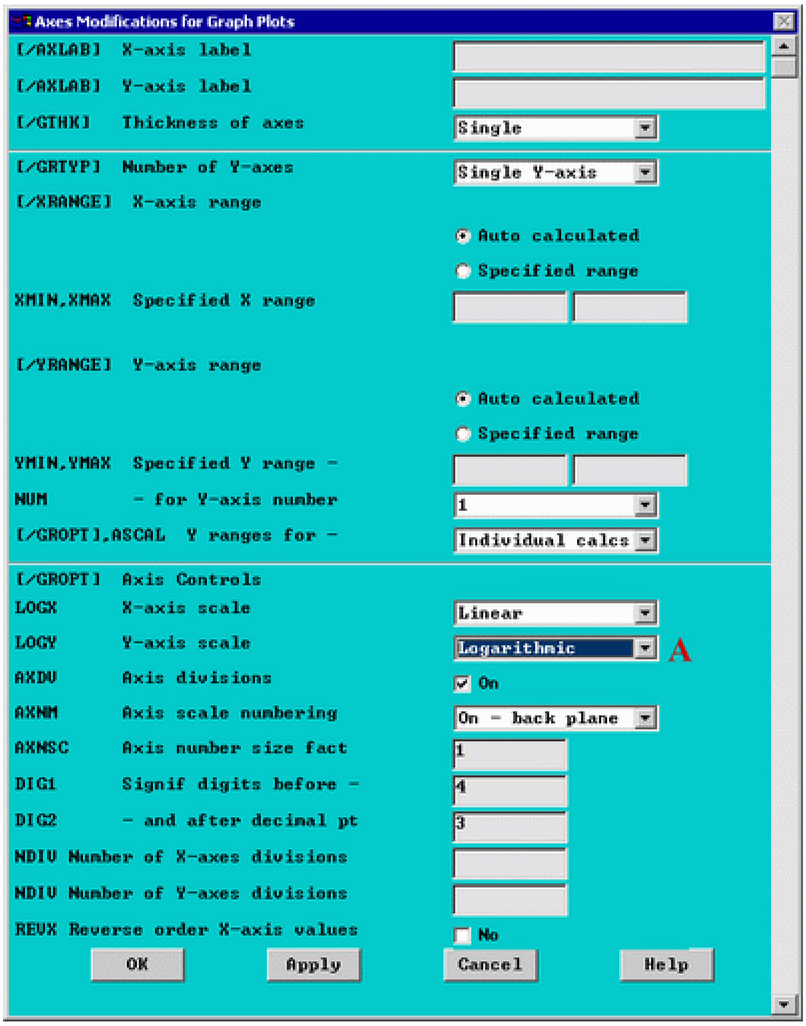


Note that we get peaks at frequencies of approximately 8.3 and 51 Hz. This corresponds with the predicted frequencies of 8.311 and 51.94Hz.

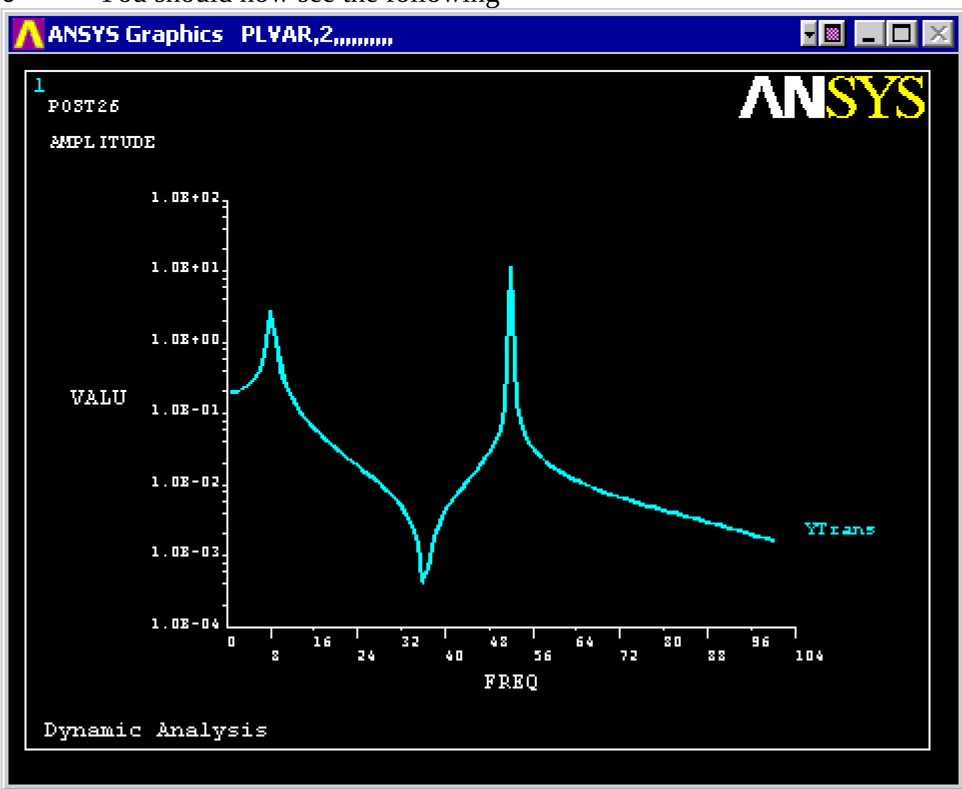
To get a better view of the response, view the log scale of **UY**.

o Select **Utility Menu > PlotCtrls > Style > Graphs > Modify Axis**

The following window will appear



- o As marked by an 'A' in the above window, change the Y-axis scale to 'Logarithmic'
- o Select **Utility Menu > Plot > Replot**
- o You should now see the following



This is the response at node 2 for the cyclic load applied at this node from 0 - 100 Hz.

- For ANSYS version lower than 7.0, the 'Variable Viewer' window is not available. Use the 'Define Variables' and 'Store Data' functions under TimeHist Postpro. See the help file for instructions.

Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to **'File > Read input from...'** and select the file.

```
FINISH
/CLEAR

/TITLE, Dynamic Analysis
/PREP7

K,1,0,0                        ! Enter keypoints
K,2,1,0

L,1,2                          ! Create line

ET,1,BEAM3                     ! Element type

R,1,0.0001,8.33e-10,0.01       ! Real Const: area,I,height

MP,EX,1,2.068e11               ! Young's modulus
MP,PRXY,1,0.33                 ! Poisson's ratio
MP,DENS,1,7830                 ! Density

LESIZE,ALL,,,10                ! Element size
LMESH,1                        ! Mesh line

FINISH
/SOLU

ANTYPE,3                       ! Harmonic analysis

DK,1,ALL                       ! Constrain keypoint 1
FK,2,FY,100                    ! Apply force

HARFRQ,0,100,                  ! Frequency range
NSUBST,100,                    ! Number of frequency steps
KBC,1                          ! Stepped loads

SOLVE
FINISH

/POST26

NSOL,2,2,U,Y, UY_2             ! Get y-deflection data
STORE,MERGE

PRVAR,2                        ! Print data
PLVAR,2                        ! Plot data
```

۸ **Reference**

http://www.mece.ualberta.ca/tutorials/ansys