ANSYS - Vibration Analysis of a Frame - Step 1

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

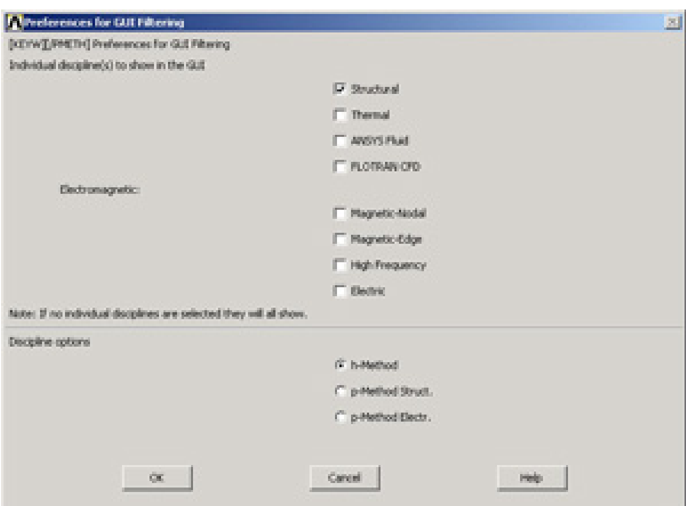
Specify directory and job name

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