ANSYS Tutorial

Slides to accompany lectures in

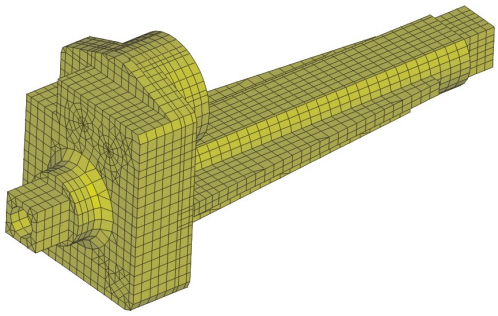
Vibro-Acoustic Design in Mechanical Systems

Department of Mechanical Engineering
University of Kentucky
Lexington, KY 40506-0503
Tel: 859-218-0609
dherrin@engr.uky.edu

Modal Analysis

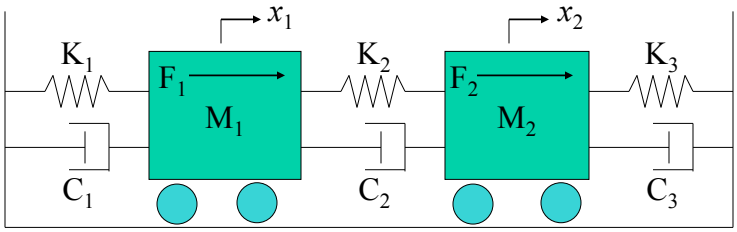
Modal/Harmonic Analysis Using ANSYS

- Used to determine the natural frequencies and mode shapes of a continuous structure



Review of Multi DOF Systems

Modal/Harmonic Analysis Using ANSYS



- Expressed in matrix form as

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{F\}$$

Review of Multi DOF Systems

Modal/Harmonic Analysis Using ANSYS

$$[M]\{\ddot{u}\}+[C]\{\dot{u}\}+[K]\{u\}=\{F\}$$

- The mass, damping and stiffness matrices are constant with time
- The unknown nodal displacements vary with time

Modal Analysis

Modal/Harmonic Analysis Using ANSYS

- A continuous structure has an infinite number of degrees of freedom
- The finite element method approximates the real structure with a finite number of DOFs
- N mode shapes can be found for a FEM having N DOFs
- Modal Analysis
  - ✓ Process for determining the N natural frequencies and mode shapes

Modal Analysis

Modal/Harmonic Analysis Using ANSYS

- Given "suitable" initial conditions, the structure will vibrate
  - ✓ at one of its natural frequencies
  - ✓ the shape of the vibration will be a scalar multiple of a mode shape
- Given "arbitrary" initial conditions, the resulting vibration will be a
  - ✓ Superposition of mode shapes

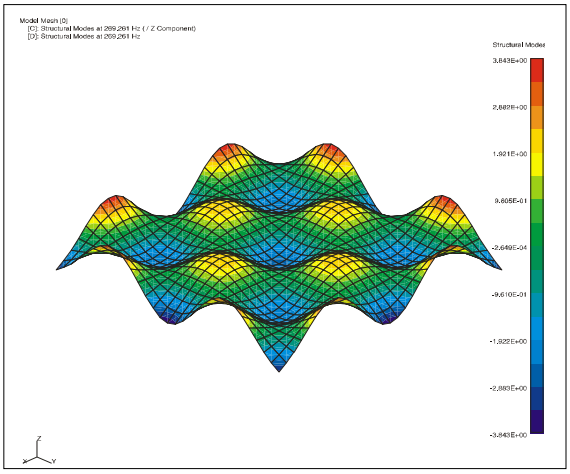
Modal Analysis

Modal/Harmonic Analysis Using ANSYS

- Determines the vibration characteristics (natural frequencies and mode shapes) of a structural components
- Natural frequencies and mode shapes are a starting point for a transient or harmonic analysis
  - ✓ If using the mode superposition method

Mode Shape of a Thin Plate (240 Hz)

Modal/Harmonic Analysis Using ANSYS



Mode Extraction Methods

Modal/Harmonic Analysis Using ANSYS

- Subspace
- Block Lanczos
- PowerDynamics
- Reduced
- Unsymmetric
- Damped and QR damped (Include damping)

Steps in Modal Analysis

Modal/Harmonic Analysis Using ANSYS

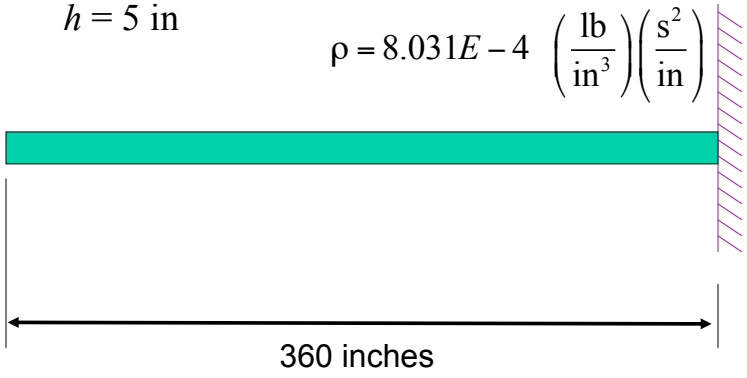
- Build the model
  - ✓ Same as for static analysis
  - ✓ Use top-down or bottom-up techniques
- Apply loads and obtain solution
  - ✓ Only valid loads are zero-value displacement constraints
  - ✓ Other loads can be specified but are ignored
- Expand the modes and review results

In-Class Exercise

Modal/Harmonic Analysis Using ANSYS

$b = 43$ in $\qquad E = 30\text{E}6$ psi

$h = 5$ in $\qquad \rho = 8.031E-4 \left(\frac{\text{lb}}{\text{in}^3}\right)\left(\frac{\text{s}^2}{\text{in}}\right)$



In-Class Exercise

Modal/Harmonic Analysis Using ANSYS

- Set element type to BEAM188
- Set the appropriate material constants and section properties
- Create Keypoints at the start and end of the beam and a Line between them

Create Three Keypoints

- Preprocessor > Modeling - Create > Keypoints > In Active CS
  - Enter the following values for keypoint 1
    - ✓ NPT=1, x=0, y=0 z=0 <Apply>
  - Enter the following values for keypoint 2
    - ✓ NPT=2, x= 180, y=0 z=0 <Apply>
  - Enter the following values for keypoint 3
    - ✓ NPT=3, x= 360, y=0 z=0 <Apply>
  - <Okay>

Create Lines Between Keypoints

- Preprocessor > Modeling - Create > Lines > Straight Line
  - Select KP 1 and 2 in graphics window
  - Select KP 2 and 3 in graphics window
  - <Okay>

Mesh the Lines to Create Elements

Modal/Harmonic Analysis Using ANSYS

- Preprocessor > Meshing - Size Controls > Lines - All Lines >
  - Size = 5
  - <Okay>
- Preprocessor > Meshing - Mesh Lines > Pick All
  - <Okay>

Mesh the Line and Apply B.C.s

Modal/Harmonic Analysis Using ANSYS

- Fix the Keypoint at the right end of the beam

Set Solution Options

Modal/Harmonic Analysis Using ANSYS

- Change the analysis type to Modal
  - ✓ Solution > Analysis Type > New Analysis
  - ✓ <Modal>
- Set the analysis options
  - ✓ Solution > Analysis Options
  - ✓ Extract 10 mode <OK>
  - ✓ Enter <1500> for the ending frequency

Set Solution Options

Modal/Harmonic Analysis Using ANSYS

- At this point, you have told ANSYS to find a particular quantity of modes and to look within a particular frequency range. If ANSYS finds that quantity before it finishes the frequency range, it will stop the search. If ANSYS does not find that quantity before finishing the frequency range, then it will stop the search.

Set Solution Options

- Solve the load set
- ANSYS generates a substep result for each natural frequency and mode shape

Postprocessing

- List results summary
  - ✓ General Postproc > List Results > Results Summary
- Read results for a substep
  - ✓ General Postproc > Read Results > First Set
  - ✓ Plot deformed geometry
  - ✓ General Postproc > Read Results > Next Set
  - ✓ Plot deformed geometry

Harmonic Response Analysis

Modal/Harmonic Analysis Using ANSYS

- Solves the time-dependent equations of motion for linear structures undergoing steady-state vibration
- All loads and displacements vary sinusoidally at the same frequency

$$F_i = F\sin(\omega t + \phi_1)$$

$$F_j = F\sin(\omega t + \phi_2)$$

Harmonic Response Analysis

Modal/Harmonic Analysis Using ANSYS

- Analyses can generate plots of displacement amplitudes at given points in the structure as a function of forcing frequency

Forced Response

Modal/Harmonic Analysis Using ANSYS

- Apply a 1.0 N load at the left end of the beam
- New Analysis > Harmonic
- Set the Analysis Options
  - ✓ Set the solution method to "Mode Superposition"
  - ✓ Set the DOF printout format to "Amplitude and phase" <OK>
  - ✓ Set the number of modes to 10 <OK>

Forced Response

Modal/Harmonic Analysis Using ANSYS

- Set the frequency substeps
  - ✓ Solution > Load Step Opts – Time Frequency > Freq and Substeps
  - ✓ Set the Harmonic Frequency Range to between 0 and 50 Hz
  - ✓ Set the number of substeps to 100
  - ✓ Set to Stepped

Forced Response

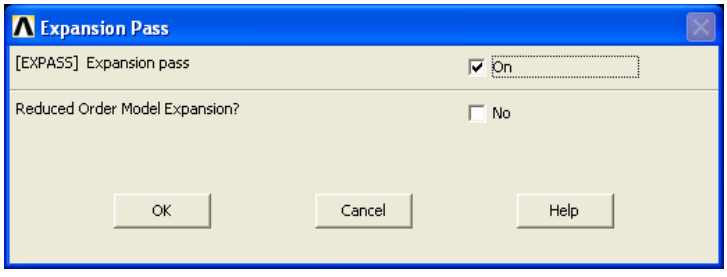
Modal/Harmonic Analysis Using ANSYS

- Set the damping
  - ✓ Solution > Load Step Opts – Time Frequency > Damping
  - ✓ Set the Constant Damping Ratio to 0.01
- Solve the model

Expansion Pass

Modal/Harmonic Analysis Using ANSYS

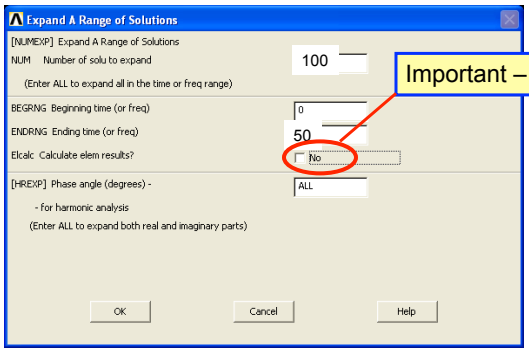
Finish
Solution > Analysis Type > Expansion Pass …



Expansion Pass Setup

Modal/Harmonic Analysis Using ANSYS

Solution > Load Step Opts > Expansion Pass >
Single Expand > Range of Solu's

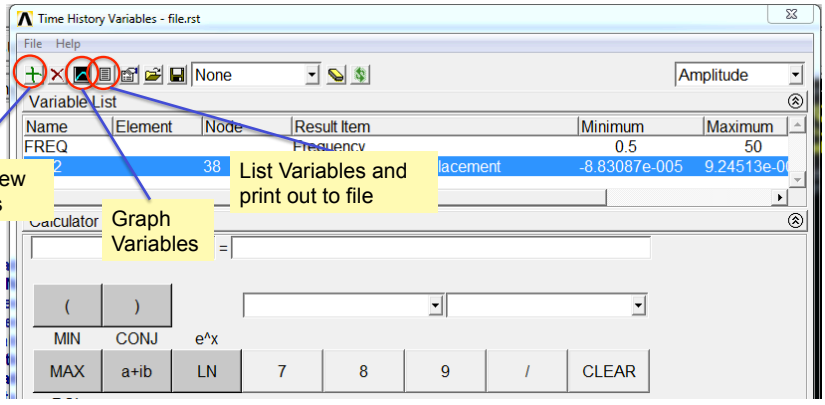


Solution > Solve > Current LS

Forced Response

Modal/Harmonic Analysis Using ANSYS

- Enter time history postprocessor > Define Variables …



Create Nodes

- Preprocessor > Modeling - Create > Nodes > In Active CS
  - Enter the following values for Node 1
    - ✓ NPT=1, x=180, y=-10 z=0 <Apply>

Add Vertical Truss Member

- Preprocessor > Element type > Add/Edit/Delete
  - <Add>
  - <Okay> Select Link 180
  - <Close>
- Preprocessor > Real Constants > Add/Edit/Delete
  - <Add>
  - <Okay> to select type 1
  - Enter the values
    - ✓ A= 10 in^2
  - <Okay>

Set Element Attributes

Modal/Harmonic Analysis Using ANSYS

- Modeling > Create > Elements > Elem Attributes
  - Select appropriate material properties and real constant table
- Preprocessor > Modeling - Create > Elements > Auto-Numbered-Thru Nodes
  - Select appropriate nodes and apply