

Course in ANSYS

Day 5

Lesson 13. Vibration/dynamic Analysis

Vibration/dynamic Analysis

Outline for Course in ANSYS:

Day 1

- Lesson 1. Introduction to ANSYS
- Lesson 2. Basics
- Lesson 3. Solution phases

Day 2

- Lesson 4. Modeling

Day 3

- Lesson 5. Material
- Lesson 6. Loading
- Lesson 7. Solution

Day 4

- Lesson 8. Structural analysis
- Lesson 9. Postprocessing
- Lesson 10. Constraint equations
- Lesson 11. Parameters
- Lesson 12. Macros

Day 5

- Lesson 13. Vibration/dynamic analysis**
- Lesson 14. Thermal

Vibration/dynamic Analysis

Programme for Lesson:

- Modal analysis (MA)
 - MA – Steps
 - MA – Input
 - MA – ExpansionPass
 - MA – Analysis Options
 - MA – Mode extraction method
 - MA – Options
 - MA – Define Loads
 - MA – Load Step Options
 - MA – General Postprocessing
- Harmonic response analysis (HRA)
 - HRA – Steps
 - HRA – Loads
 - HRA – Input
 - HRA – Load Step Options
 - HRA – Freq and Substps
 - HRA – Solution options
- Transient dynamic analysis (TDA)
 - TDA – Steps
 - TDA – Solution Methods
 - TDA – Analysis options
 - TDA – Solution Controls
- Spectrum analysis (SA)
 - Steps in a Single-Point Response Spectrum (SPRS) Analysis
 - Steps in Random Vibration (PSD) Analysis

Lesson 13

3

Modal analysis (MA)

- You use modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.

Lesson 13

4

Modal analysis (MA)

- You use modal analysis to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required if you want to do a spectrum analysis or a mode superposition harmonic or transient analysis.

Lesson 13

5

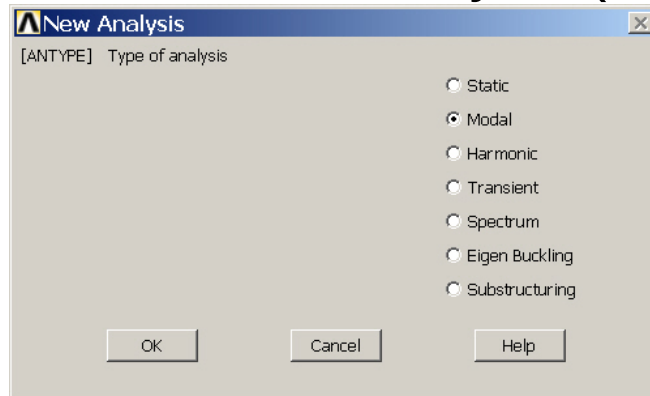
Modal analysis (MA)

- You can do modal analysis on a prestressed structure, such as a spinning turbine blade.
- Modal analysis in the ANSYS family of products is a linear analysis.
- Extraction methods:
 - Block Lanczos (default)
 - Subspace
 - PowerDynamics
 - Reduced
 - Unsymmetric
 - Damped
 - QR damped
- The damped and QR damped methods allow you to include damping in the structure.
- Restarts are not valid in a modal analysis. If you need to apply different sets of boundary conditions, do a new analysis each time

Lesson 13

6

Modal analysis (MA)



Lesson 13

7

MA - Steps

- Build the model.
- Setup the solution.
- Apply loads.
- Run the solution.
- Expand the modes.
- Review the results.

Lesson 13

8

MA - Input

BUILD THE MODEL

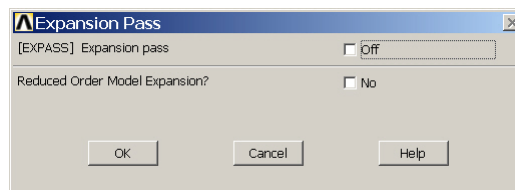
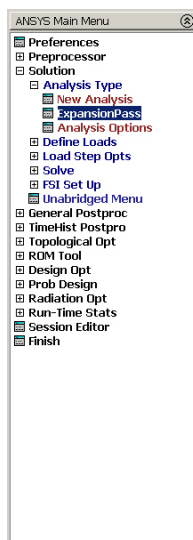
- Only linear behavior is valid in a modal analysis. Nonlinear input are ignored.
- Define
 - Young's modulus (EX) (or stiffness in some form)
 - Density (DENS) (or mass in some form)

Lesson 13

9

MA - ExpansionPass

SETUP THE SOLUTION



writing mode shapes to the results file

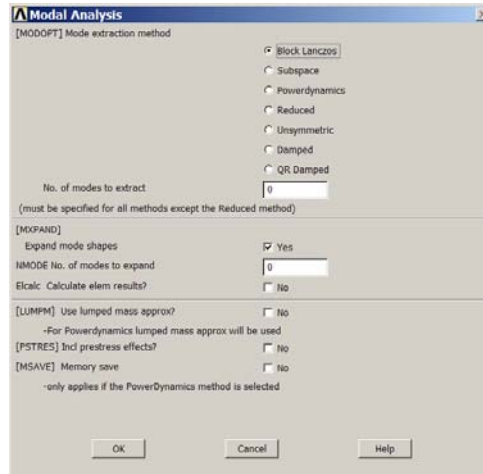
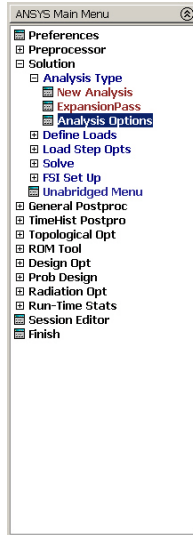
You must explicitly leave SOLUTION (using the **FINISH** command) and re-enter (**/SOLU**) before performing the expansion pass.

Lesson 13

10

MA - Analysis Options

SETUP THE SOLUTION



Lesson 13

11

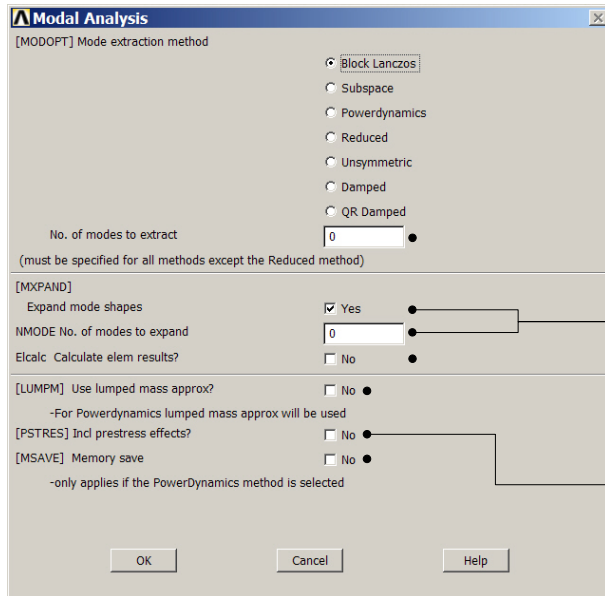
MA - Mode extraction method

SETUP THE SOLUTION

- Block Lanczos (default) method
 - used for large symmetric eigenvalue problems
 - uses the sparse matrix solver, overriding any solver specified via the [EQSLV](#) command
 - a faster convergence rate achieved compared to the Subspace method
- Subspace method
 - for large symmetric eigenvalue problems
- PowerDynamics method
 - used for very large models (100,000+ DOFs)
 - automatically uses the lumped mass approximation
- Reduced (Householder) method
 - uses reduced (condensed) system matrices to calculate the solution
 - less accurate because the reduced mass matrix is approximate
 - need to define master degrees of freedom
- Unsymmetric method
 - used for problems with unsymmetric matrices, such as fluid-structure interaction problems
- Damped method
 - used for problems where damping cannot be ignored, such as bearing problems
- QR damped
 - uses the reduced modal damped matrix to calculate complex damped frequencies in modal coordinates
 - faster and achieves better calculation efficiency compared to the damped method

Lesson 13

12



MA - Options

SETUP THE SOLUTION

Number of modes to expand and write. If blank, expand and write all modes within the frequency range specified.

Use this option to calculate the modes of a prestressed structure.

Lesson 13

13

MA - Define Loads

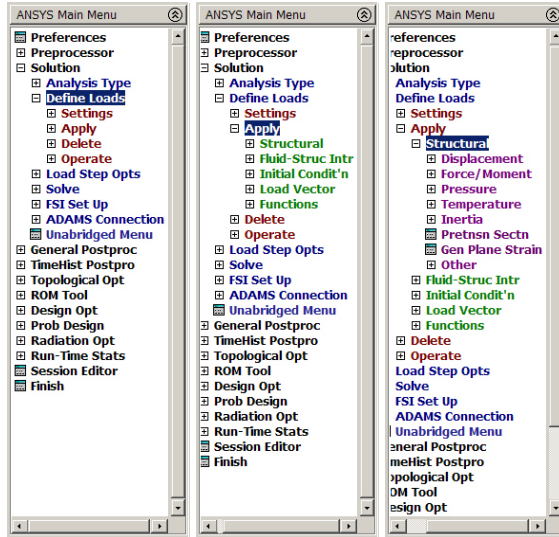
APPLY LOADS

- The only "loads" valid in a typical modal analysis are zero-value displacement constraints.
 - If you input a nonzero displacement constraint, the program assigns a zero-value constraint to that DOF instead.
- Other loads can be specified, but are ignored.

Lesson 13

14

MA - Define Loads



APPLY LOADS

Lesson 13

15

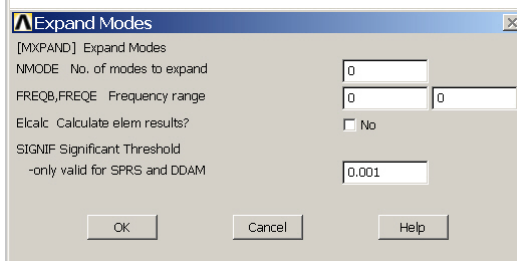
MA - Load Step Options



EXPAND THE MODES

Only expanded modes can be reviewed in the postprocessor.

Default is no modes expanded.



Lesson 13

16

MA - General Postprocessing

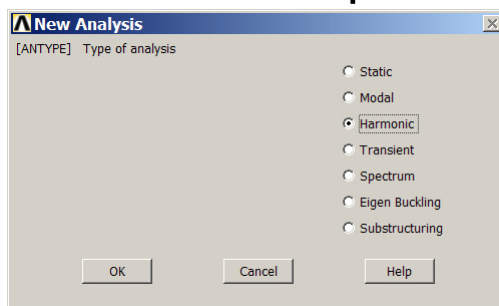
REVIEW THE RESULTS

- The results file (Jobname.RST) must be available.
- Read in results data from the appropriate substep.
- Each mode is stored on the results file as a separate substep.
 - If you expand six modes, for instance, your results file will have one load step consisting of six substeps.

Lesson 13

17

Harmonic response analysis (HRA)



Lesson 13

18

Harmonic response analysis (HRA)

- *Harmonic response analysis* gives you the ability to predict the sustained dynamic behavior of your structures, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

Lesson 13

19

Harmonic response analysis (HRA)

- Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidally (*harmonically*) with time.

Lesson 13

20

Harmonic response analysis (HRA)

- The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies.
- This analysis technique calculates only the steady-state, forced vibrations of a structure.

Lesson 13

21

Harmonic response analysis (HRA)

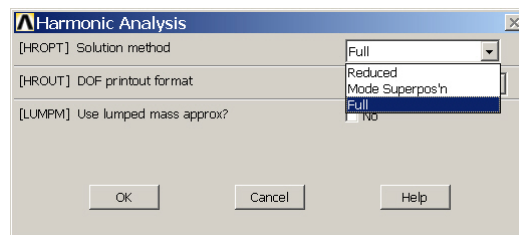
- Linear
- Plasticity is ignored
- Fluid-structure interaction can be handled
- Prestress can be included

Lesson 13

22

Harmonic response analysis (HRA)

- The Full Method
- The Reduced Method
- The Mode Superposition Method



Lesson 13

23

HRA - Steps

- Build the model. (Similar to MA)
- Apply loads.
- Run the solution.
- Review the results.

Lesson 13

24

HRA - Loads

APPLY LOADS

- The applied load varies harmonically (sinusoidally) with time
 - *amplitude*
 - *phase angle*
 - *forcing frequency range*

Lesson 13

25

HRA - Input

APPLY LOADS

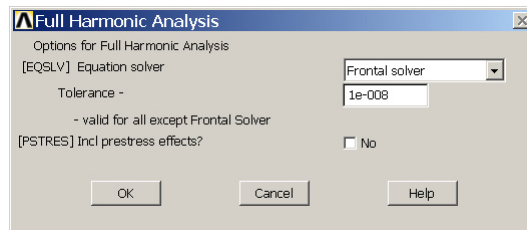
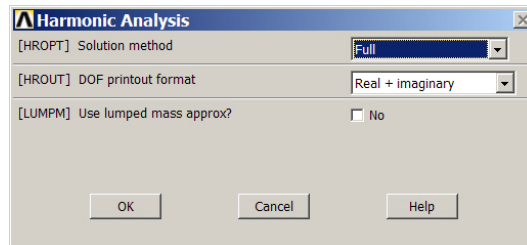
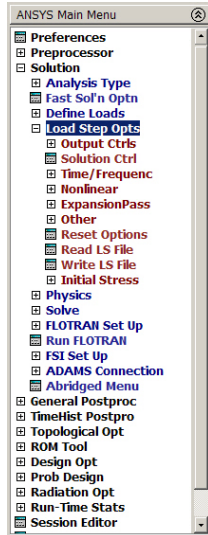
- Only linear behavior is valid in a harmonic response analysis. Nonlinear elements, if any, will be treated as linear elements. If you include contact elements, for example, their stiffnesses are calculated based on their initial status and are never changed.
- Both Young's modulus (EX) (or stiffness in some form) and density (DENS) (or mass in some form) must be defined. Material properties may be linear, isotropic or orthotropic, and constant or temperature-dependent. Nonlinear material properties, if any, are ignored.

Lesson 13

26

HRA – Load Step Opts

APPLY LOADS

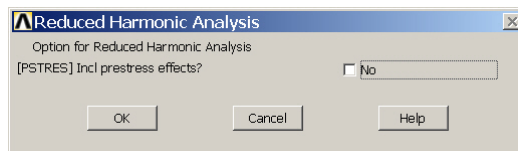
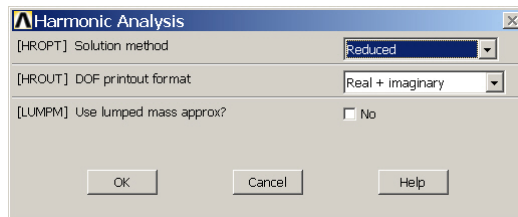
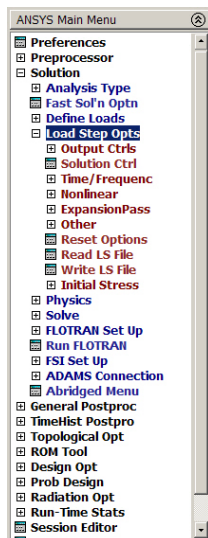


Lesson 13

27

HRA – Load Step Opts

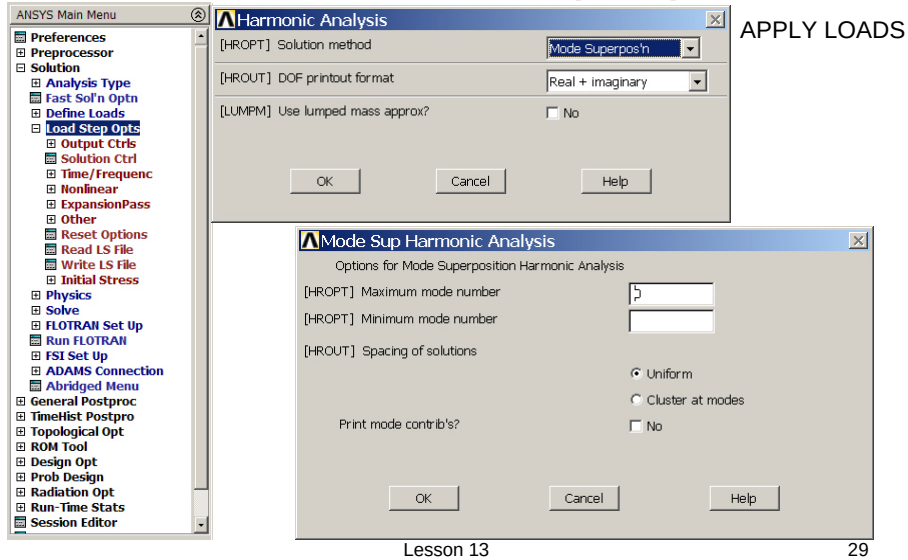
APPLY LOADS



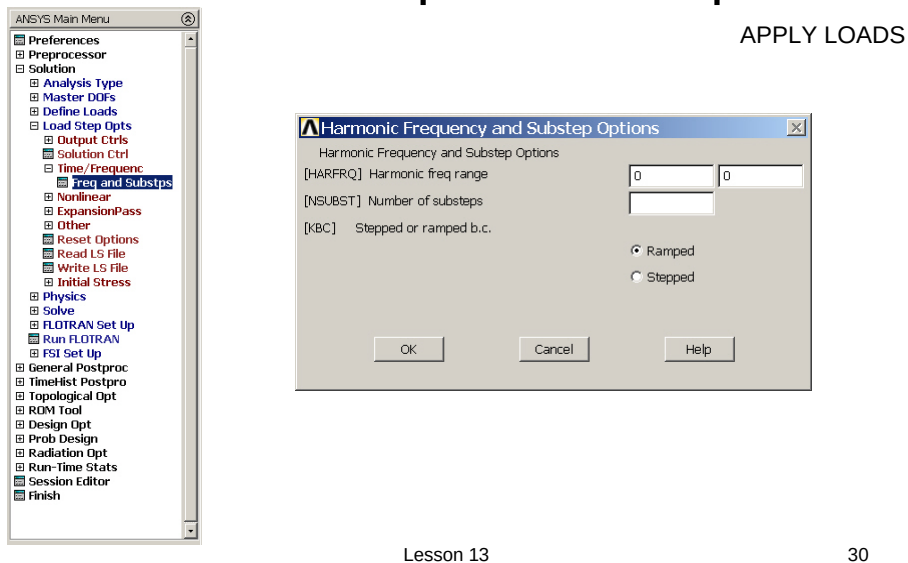
Lesson 13

28

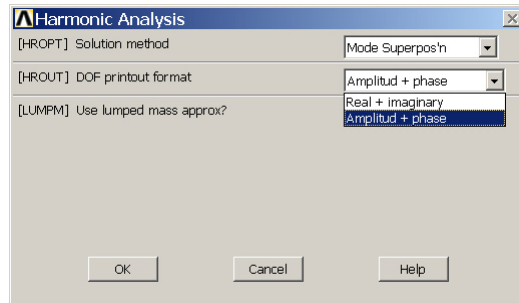
HRA – Load Step Opts



HRA – Freq and Substps



HRA – Solution options

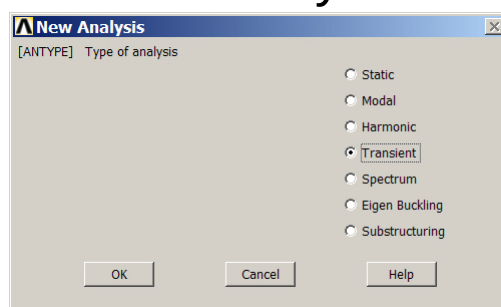


RUN THE SOLUTION

Lesson 13

31

Transient dynamic analysis (TDA)



Lesson 13

32

Transient dynamic analysis (TDA)

- Transient dynamic analysis (sometimes called time-history analysis) is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads.
- You can use this type of analysis to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any combination of static, transient, and harmonic loads.
- The time scale of the loading is such that the inertia or damping effects are considered to be important.

Lesson 13

33

Transient dynamic analysis (TDA)

- A transient dynamic analysis is more involved than a static analysis because it generally requires more computer resources and more of *your* resources, in terms of the “engineering” time involved. You can save a significant amount of these resources by doing some preliminary work to understand the physics of the problem. For example, you can:
 - Analyze a simpler model first. A model of beams, masses, and springs can provide good insight into the problem at minimal cost. This simpler model may be all you need to determine the dynamic response of the structure.
 - If you are including nonlinearities, try to understand how they affect the structure's response by doing a static analysis first. In some cases, nonlinearities need not be included in the dynamic analysis.
 - Understand the dynamics of the problem. By doing a modal analysis, which calculates the natural frequencies and mode shapes, you can learn how the structure responds when those modes are excited. The natural frequencies are also useful for calculating the correct integration time step.
 - For a nonlinear problem, consider substructuring the linear portions of the model to reduce analysis costs.

Lesson 13

34

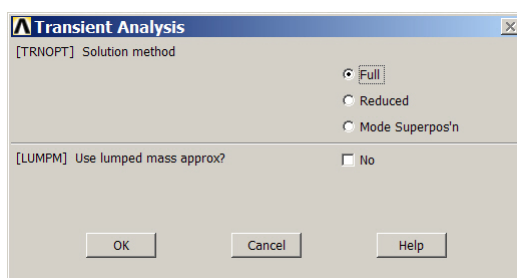
TDA - Steps

- The procedure for a full transient dynamic analysis (available in the ANSYS Multiphysics, ANSYS Mechanical, and ANSYS Structural products) consists of these steps:
 - Build the Model
 - Establish Initial Conditions
 - Set Solution Controls
 - Set Additional Solution Options
 - Apply the Loads
 - Save the Load Configuration for the Current Load Step
 - Repeat Steps 3-6 for Each Load Step
 - Save a Backup Copy of the Database
 - Start the Transient Solution
 - Exit the Solution Processor
 - Review the Results

Lesson 13

35

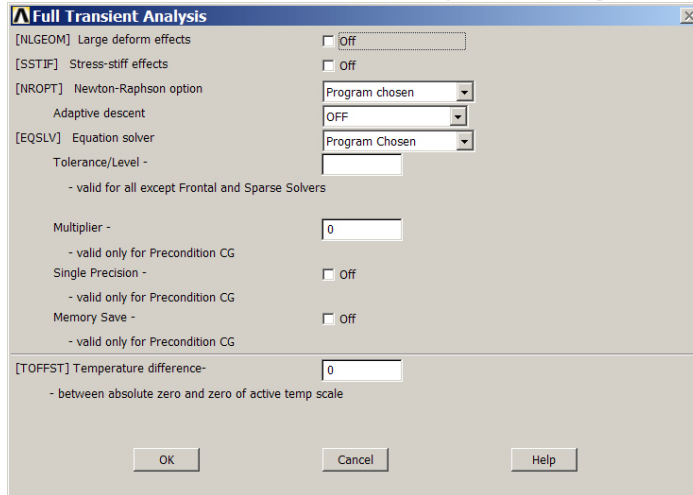
TDA – Solution Methods



Lesson 13

36

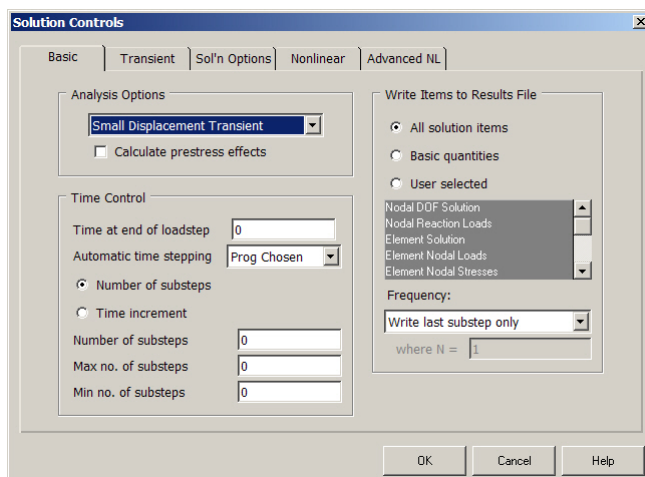
TDA – Analysis options



Lesson 13

37

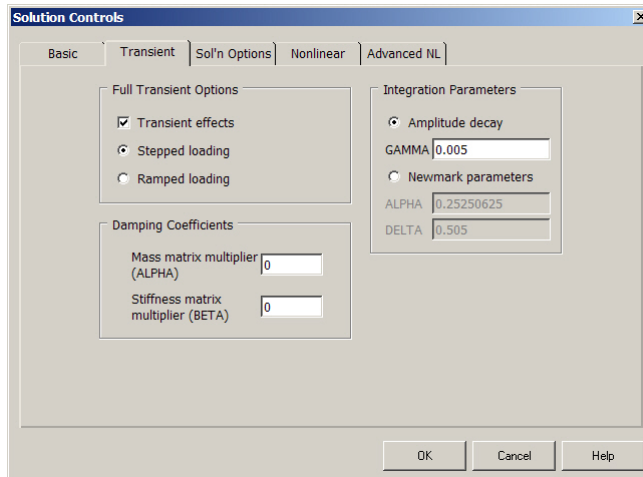
TDA – Solution Controls - Basic



Lesson 13

38

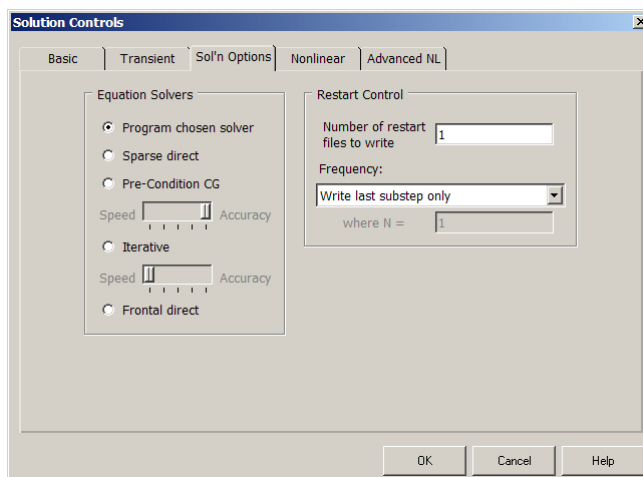
TDA – Solution Controls - Transient



Lesson 13

39

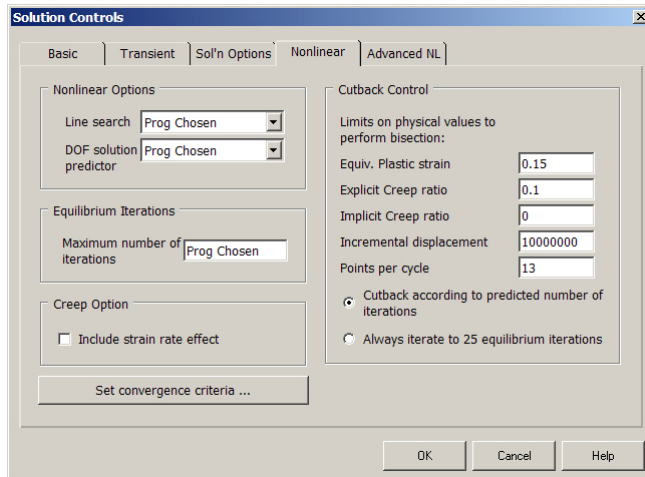
TDA – Sol. Controls - Sol'n Options



Lesson 13

40

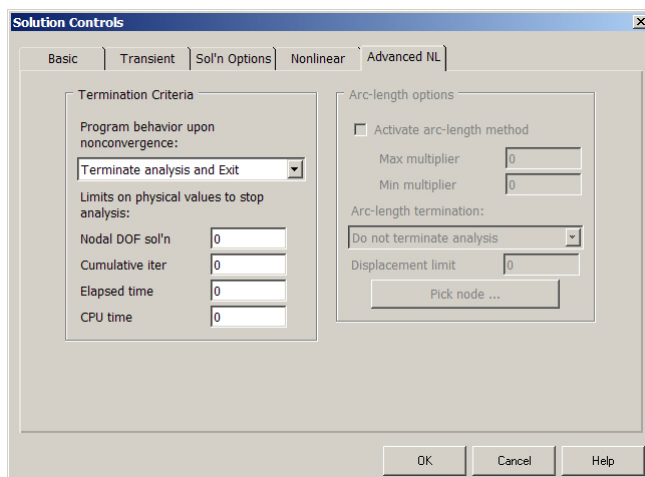
TDA – Sol. Controls - Nonlinear



Lesson 13

41

TDA – Sol. Controls - Advanced NL



Lesson 13

42

Spectrum analysis (SA)

- A spectrum analysis is one in which the results of a modal analysis are used with a known *spectrum* to calculate displacements and stresses in the model.
- It is mainly used in place of a time-history analysis to determine the response of structures to random or time-dependent loading conditions such as earthquakes, wind loads, ocean wave loads, jet engine thrust, rocket motor vibrations, and so on.

Lesson 13

43

Steps in a Single-Point Response Spectrum (SPRS) Analysis

- The procedure for a single-point response spectrum analysis consists of six main steps:
 - Build the model.
 - Obtain the modal solution.
 - Obtain the spectrum solution.
 - Expand the modes.
 - Combine the modes.
 - Review the results.

Lesson 13

44

Steps in Random Vibration (PSD) Analysis

- The procedure for a PSD analysis consists of six main steps:
 - Build the model.
 - Obtain the modal solution.
 - Expand the modes.
 - Obtain the spectrum solution.
 - Combine the modes.
 - Review the results.

Lesson 13

45

Steps in DDAM Spectrum Analysis

- The procedure for a DDAM spectrum analysis is the same as that for a single-point response spectrum (SPRS) analysis (including file requirements), with the following exceptions:
 - Use the British system of units [*inches* (not *feet*), *pounds*, etc.] for all input data - model geometry, material properties, element real constants, etc.
 - Choose DDAM instead of SPRS as the spectrum type [[SPOPT](#) command].
 - Use the [ADDAM](#) and [VDDAM](#) commands instead of [SVTYP](#), [SV](#), and [FREQ](#) to specify the spectrum values and types. Specify the global direction of excitation using the [SED](#) command. Based on the coefficients specified in the [ADDAM](#) and [VDDAM](#) commands, the program computes the mode coefficients according to the empirical equations given in the [ANSYS, Inc. Theory Reference](#).
 - The most applicable mode combination method is the NRL sum method [[NRLSUM](#)]. Mode combinations are done in the same manner as for a single-point response spectrum. Mode combinations require damping.
 - No damping needs to be specified for solution because it is implied by the [ADDAM](#) and [VDDAM](#) commands. If damping is specified, it is used for mode combinations but ignored for solution.

Lesson 13

46