Module 05: Linear Perturbation Analysis

Release 2022 R2

Please note:

- These training materials were developed and tested in Ansys Release 2022 R2. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

Module 05 Learning Outcomes

- After completing this module, you will:
  - Know the fundamental principles of Linear Perturbation.
  - Know how to include non-linear pre-stress effects within a downstream dynamic analysis.

Module 05 Topics

A. What is Linear Perturbation Analysis?

B. Modal Analysis Based on Linear Perturbation

- Setup
- Restart to Generate Tangent Stiffness Matrix
- Contact Stiffness (Status)
- Load Management
- Update Geometry Based Upon Base Analysis
- Solve the Linearly Perturbed Modal Analysis
- A Note About Post Processing

A. What is Linear Perturbation Analysis?

- In many engineering applications, the linear behavior of a structure based upon prior linear or nonlinear (static or transient) preload conditions is of interest.
- Mechanical leverages the power of linear perturbation technology for all pre-stress analyses performed within Mechanical in order to account for these prior effects.
- Large deflection static analysis followed by pre-stress modal analysis is allowed, as well as linear perturbation buckling, full harmonic, mode superposition transient, harmonic, response spectrum and random vibration, and lastly substructure analyses.
- The base static analysis can be linear, or it can include nonlinear contact, materials, and/or large deflection effects.

... What is Linear Perturbation Analysis?

- The linear perturbation analysis is essentially an iteration on top of a base (or prior) linear or nonlinear analysis.
- The perturbation analysis "freezes" the effects from the base analysis for reuse in the current or future analysis scenarios. Affected items include:
  - Solution matrices
  - Material properties
- Internally, perturbation makes use of Ansys restart capabilities in order to automate the entire process. Restart capabilities are a native feature within Mechanical.

... What is Linear Perturbation Analysis?

- In flowchart form, the solution process looks like this:

Base Analysis

- Linear or Nonlinear
- Static or Transient

Perturbation Phase I

- Restart Analysis
- Calculate matrices and material properties
- Manage loads inherited from base analysis

Perturbation Phase II

- Allow Load Modification
- Update geometry based upon base analysis
- Solve current perturbation analysis type

- Specifics of each phase will vary depending upon analysis type

... What is Linear Perturbation Analysis?

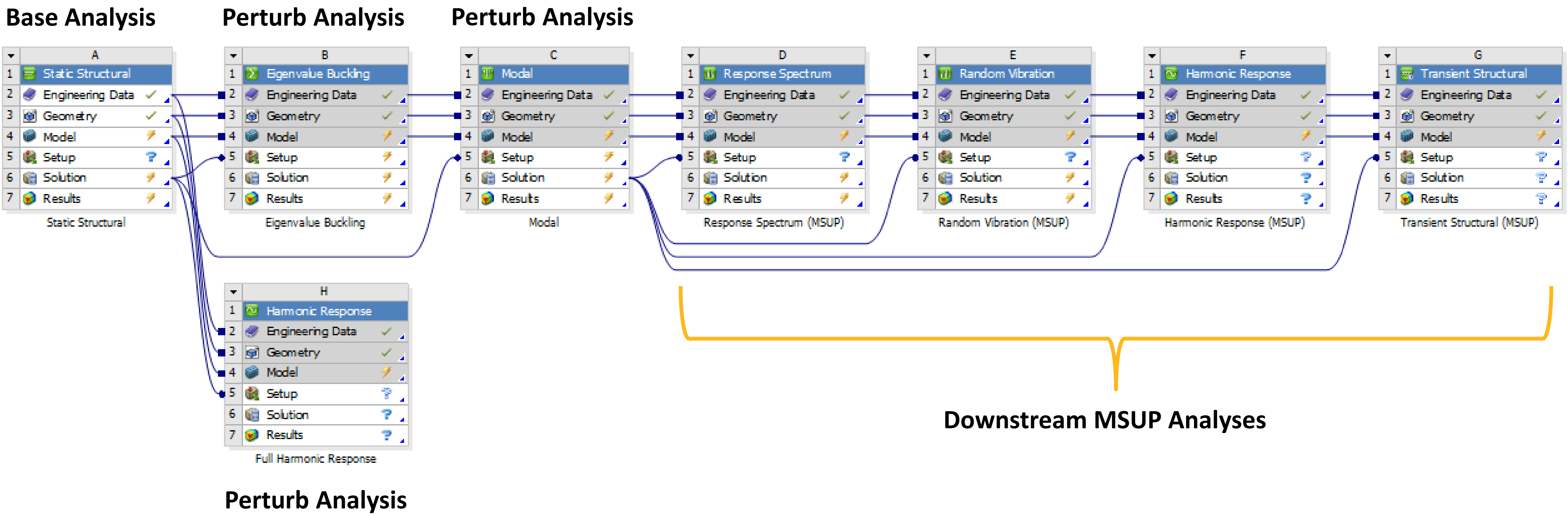
- Two key points are of fundamental importance for carrying out a linear perturbation analysis:

  1. The total tangent stiffness matrix from the prior solution (the base analysis) must be obtained for the current linear perturbation analysis. This matrix is regenerated in the first phase of the linear perturbation procedure.

  2. The total perturbation loads must be established. This load vector is calculated in the second phase of the linear perturbation procedure. This step is mandatory for future downstream mode superposition analyses (i.e. harmonic, PSD) but is not necessary for modal analysis.

… What is Linear Perturbation Analysis?

- From a workflow standpoint, the Workbench Project Schematic might look like this, accounting for all analysis possibilities with the exception of substructure analysis.

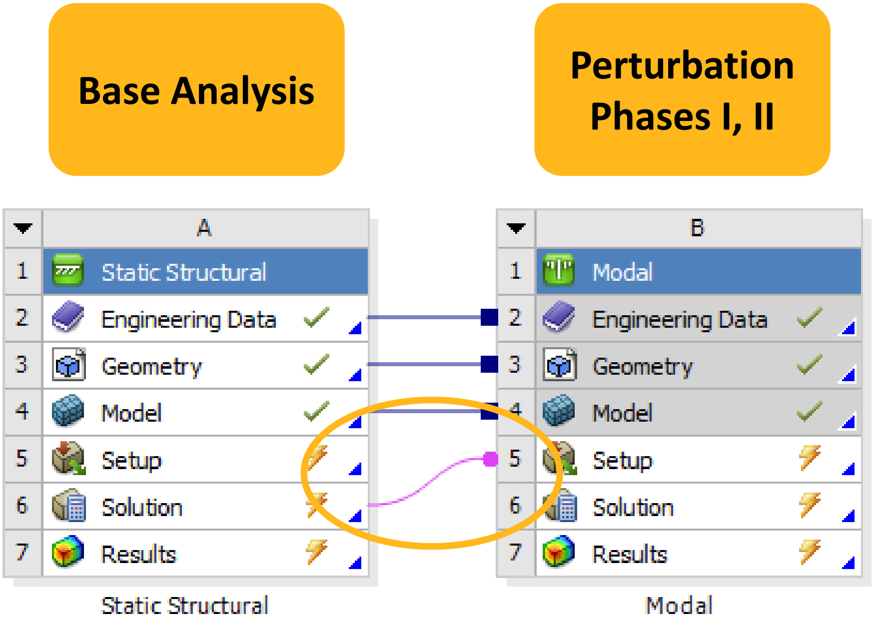


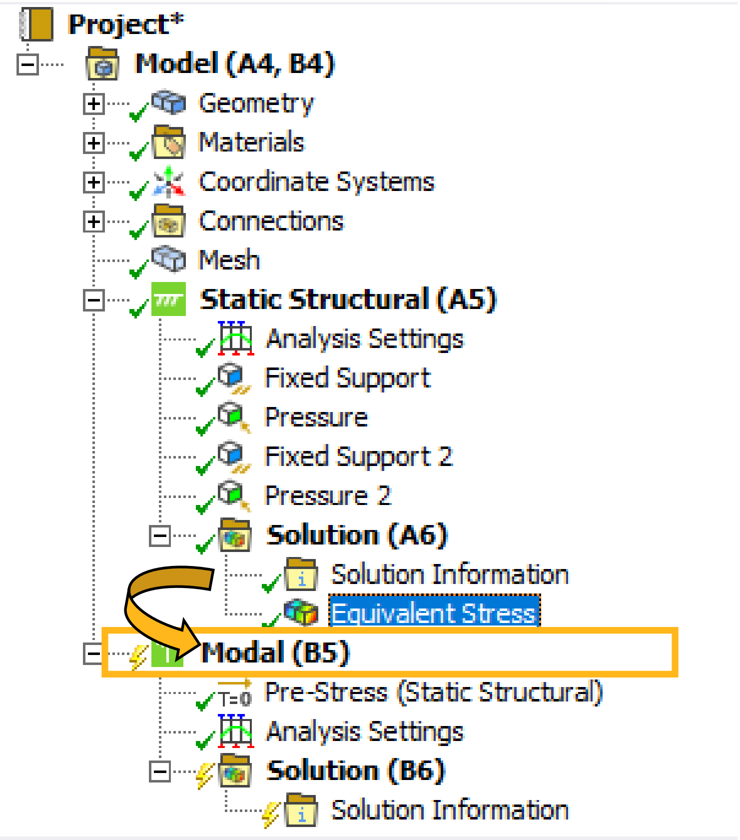
B. Modal Analysis Based on Linear Perturbation

- We will focus on modal analysis, however details of the perturbation method for other analyses can be found here: https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v222/en/ans_str/strlinpertproc.html
- A pre-stressed modal analysis can be used to calculate the frequencies and mode shapes of a pre-stressed structure, such as a spinning turbine blade, pre-stressed guitar strings, etc.
- The pre-stress influences the stiffness of the structure through a number of different stiffness contributions, including material, contact, and stress stiffening.

… Modal Analysis Based on Linear Perturbation

1. Setup
   - Linked-Modal Analysis (Pre-Stress)





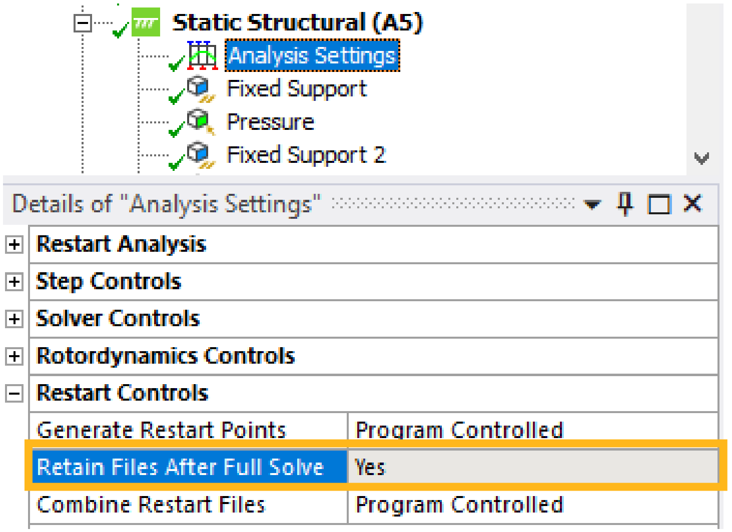
Establishing this link (A6 > B5) prior to solving the Static Structural analysis will ensure that all the necessary static conditions will be available for the subsequent perturbation solution.

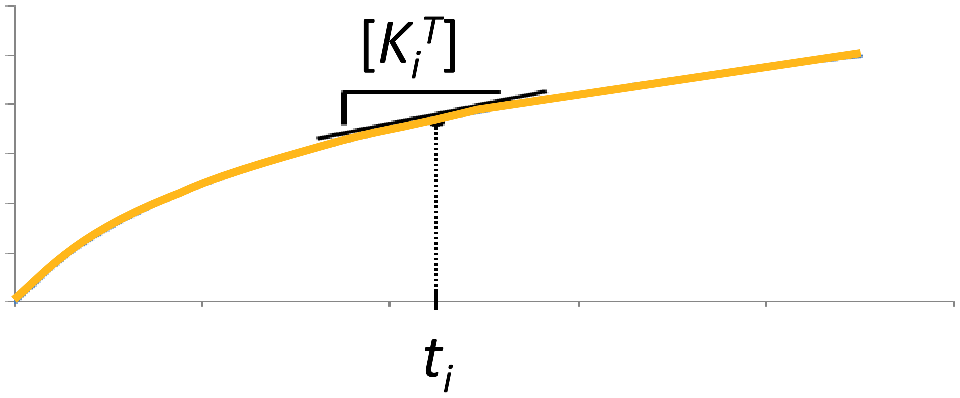
… Modal Analysis Based on Linear Perturbation

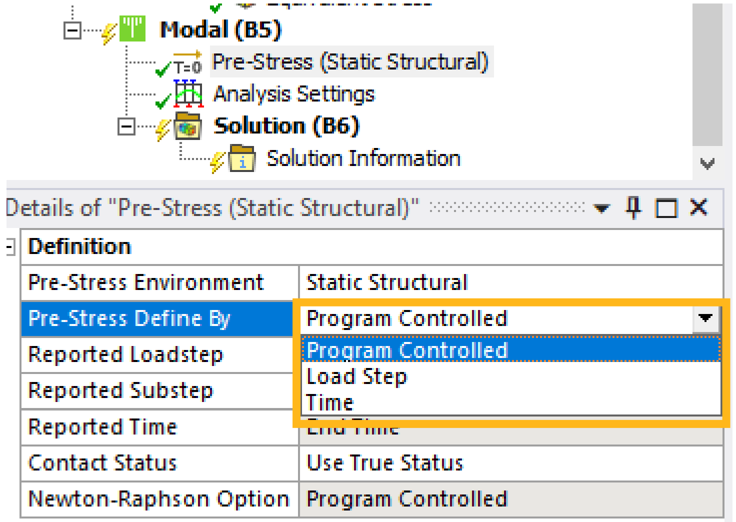
Perturbation Phase I

2. Restart to Generate Tangent Stiffness Matrix

- Restart files must be available from the base analysis. Linking Static Structural Solution and Modal Setup cells accomplishes this.
- Uses global tangent stiffness matrix $[K_i^T]$ from desired time point.
- Any time point from a static or transient structural analysis can be perturbed.



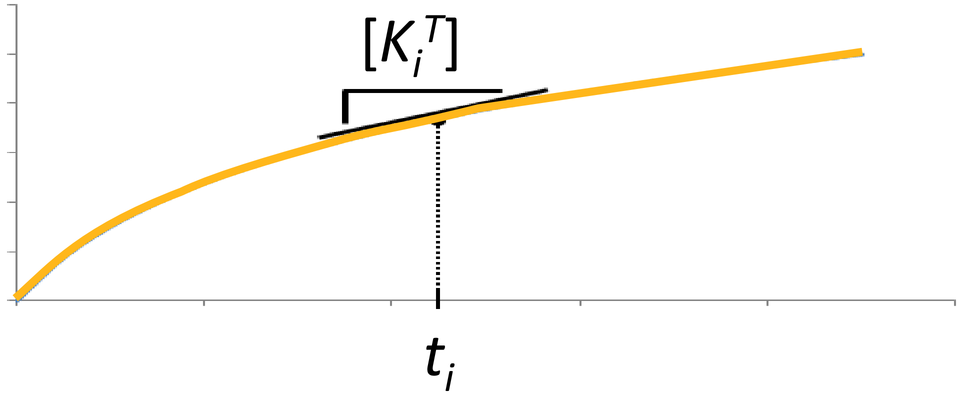




… Modal Analysis Based on Linear Perturbation

Perturbation Phase I

… Restart to Generate Tangent Stiffness Matrix

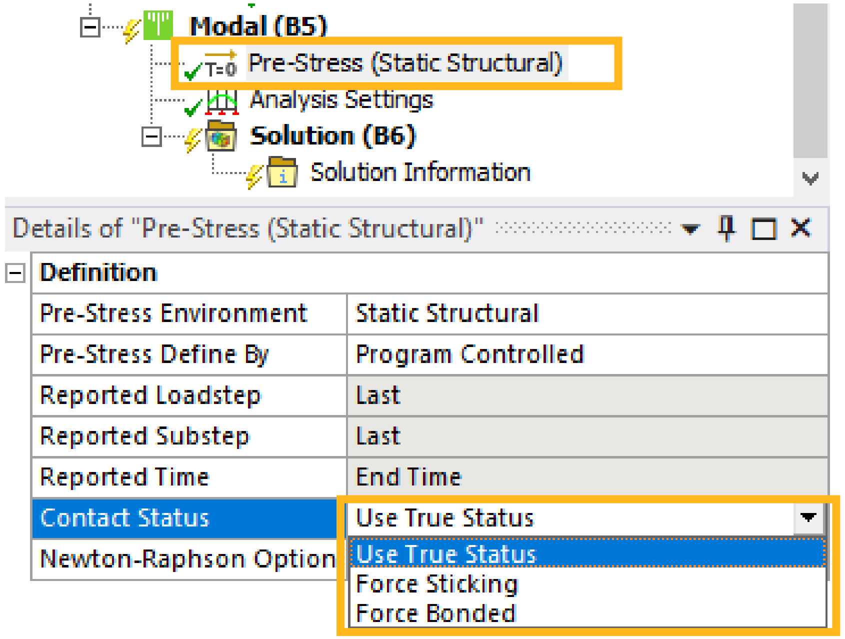


- Tangent Stiffness matrix
  - $K_i^T = K_i^{Mat} + K_i^{Contact} + K_i^{LoadStiffness} + K_i^{SpinSoftening} + K_i^{StressStiffness}$
- Material Stiffness [K_i^{Mat}]:
  - For nonlinear materials, only the linear portion is used
  - For hyperelastic materials, the tangent material properties at the point of restart are used
- Stress Stiffening [$K_i^{StressStiffness}$] / Spin softening [$K_i^{SpinSoftening}$]:
  - Effects included automatically in large deflection analyses
- Contact stiffness [$K_i^{Contact}$]:
  - Contact behavior can be changed prior to the modal analysis…

… Modal Analysis Based on Linear Perturbation

Perturbation Phase I

3. Contact Stiffness (Status)



- *__Use True Status (default):__*
  - Uses the current contact status from the restart snapshot.
  - Nonlinear contact status at the point of restart is *frozen* and used throughout the linear perturbation analysis.
- *__Force Sticking:__*
  - Uses sticking contact stiffness for the frictional contact pairs, even when the status is sliding.
  - Applies only to contact pairs whose frictional coefficient is greater than zero.
- *__Force Bonded:__*
  - For contact pairs that are in the closed (sticking/sliding) state.

Ansys

… Modal Analysis Based on Linear Perturbation

Perturbation Phase I

4. Load Management

- For linear perturbation modal analysis, all loads and constraints are deleted from the base analysis.
- All displacement constraints are kept for convenience in conducting the perturbation analysis.
- Any non-zero displacement constraints are set to zero.

… Modal Analysis Based on Linear Perturbation

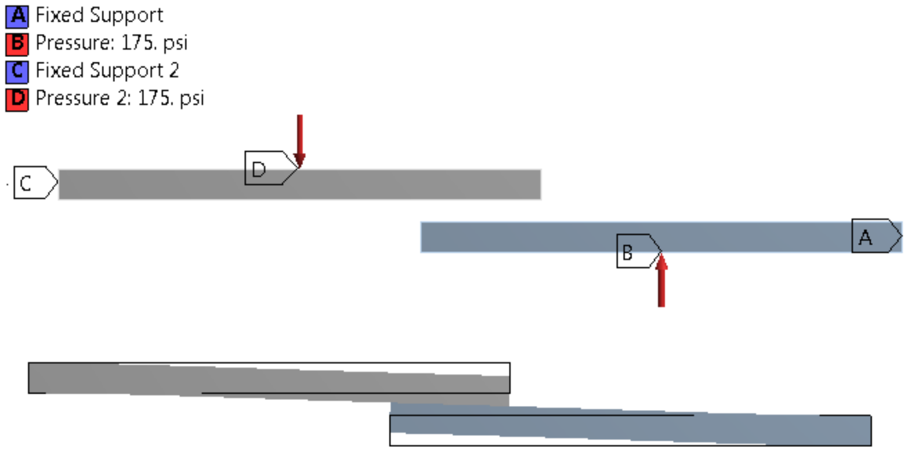
Perturbation Phase II

5. Update Geometry Based Upon Base Analysis

- This occurs automatically provided that large deflection effects are turned on in the base analysis.
- If performing a pre-stress modal analysis, it is recommended to include large deflection effects to produce accurate results in the modal analysis.

Details of "Analysis Settings"

Restart Analysis	
Step Controls	
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off



Base analysis geometry

Updated geometry at start of modal analysis

… Modal Analysis Based on Linear Perturbation

Perturbation Phase II

6. Solve the Linearly Perturbed Modal Analysis

- With $[K]^t$ already formed during Phase 1, Mechanical will continue to construct the [M] matrix and the [C] matrix (if performing damped modal analysis)
- Solve the modal analysis

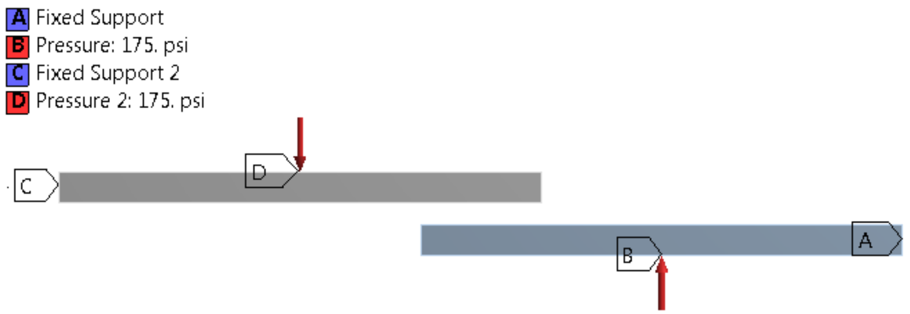
… Modal Analysis Based on Linear Perturbation

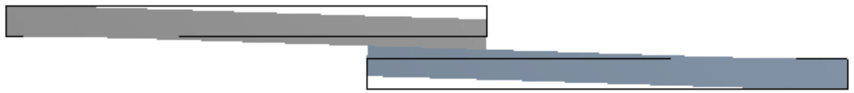
Perturbation Phase II

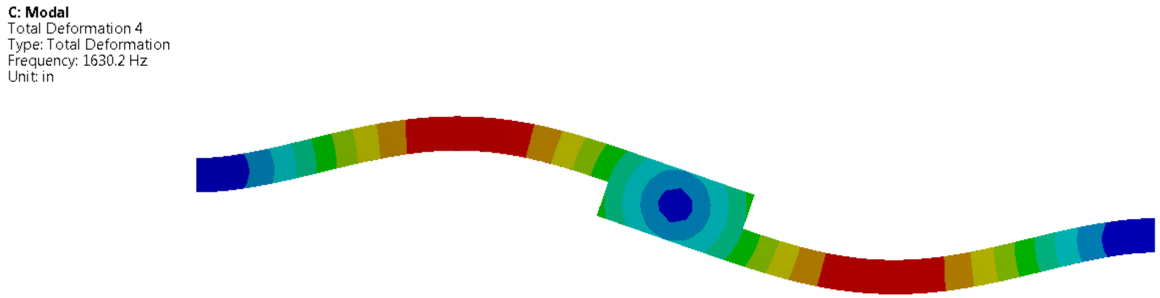
7. A Note About Post Processing

- Although the modal results (including displacements, stresses, and strains) will be correctly calculated in the modal analysis, the deformed shape picture inside Mechanical will be based upon the initial geometry from the base analysis, not the deformed geometry. If you desire to see the mode shapes based upon the deformed geometry, you can take the results into Mechanical APDL or use the Mesh Source (Beta) feature.
- The workshop following this module provides a Command Object that will enable visualization of the mode shapes inside Mechanical based upon the deformed geometry from the base analysis. The alternative Beta feature option is detailed as well.

Workshop 05.1: Linear Perturbation with Two Beams







End of presentation