Ansys Mechanical Linear and Nonlinear Dynamics

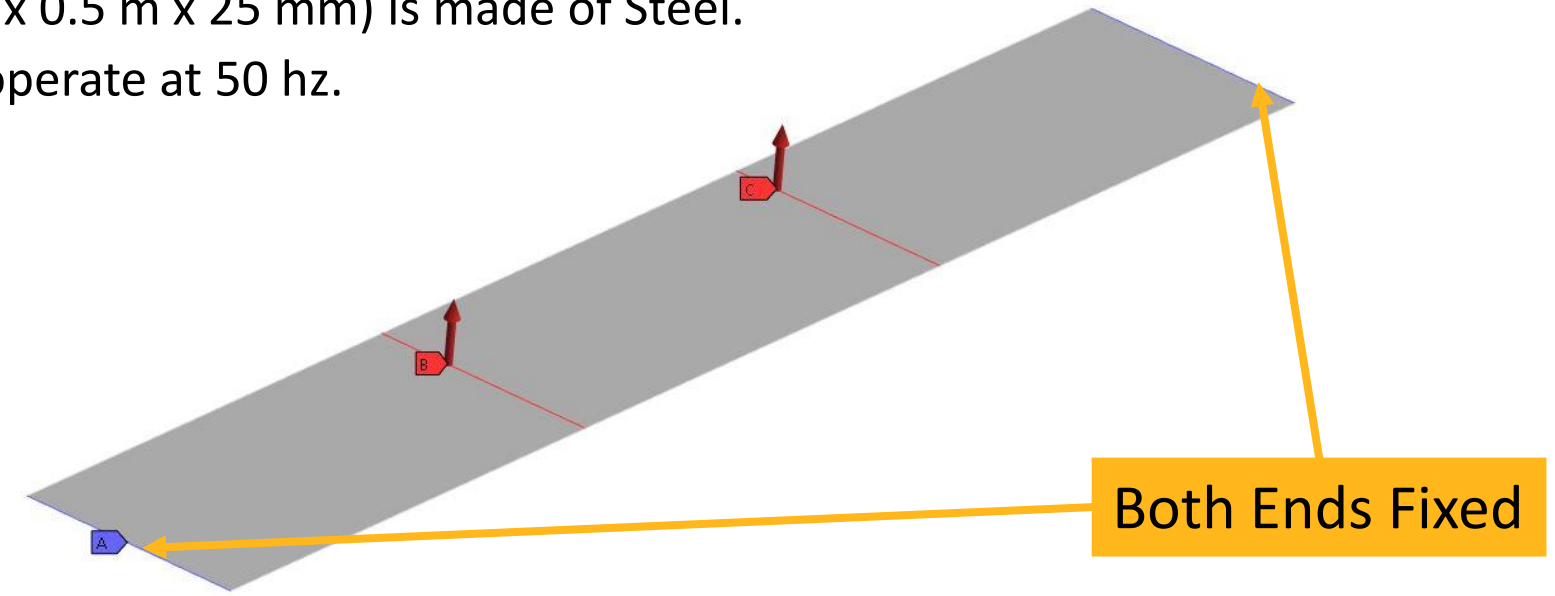
WS 06.1: Fixed-Fixed Beam

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.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

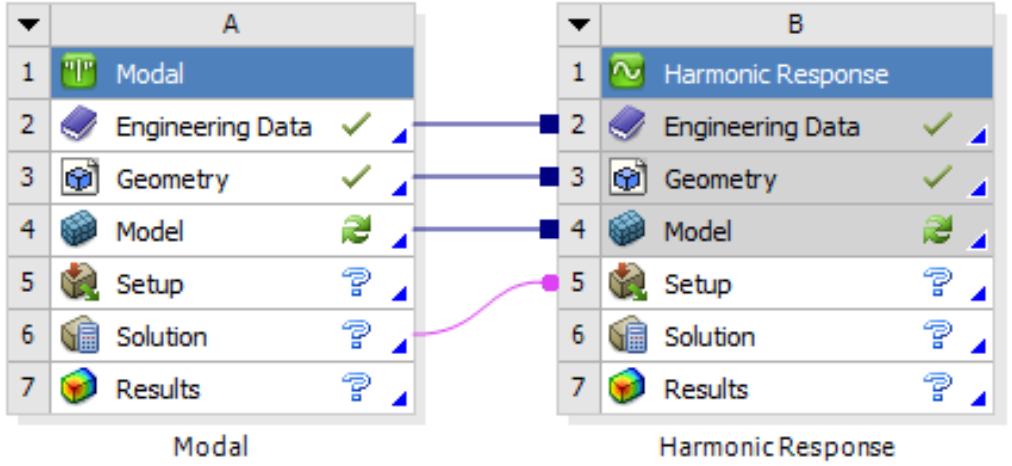
Workshop 06.1 - Goals

- Determine the harmonic response of a fixed-fixed beam under the influence of two harmonic loads.
  - Compare the Mode-Superposition method to the Full solution method.
  - Investigate the effects of damping and phase angle on the solution.
- The forces represent rotating machines mounted at the “one-third” points along the beam.
  - The beam (3 m x 0.5 m x 25 mm) is made of Steel.
  - The machines operate at 50 hz.



Workshop 06.1 - Project Schematic

- Start a new Workbench session and from the project schematic, insert a new Modal analysis system.
  - We will first look at the natural frequencies and mode shapes of the system
- Drop a Harmonic Response analysis system onto the Solution cell of the Modal system to share the material properties, geometry, mesh and solution.
  - Note that this harmonic response system will use the modes from the Modal system via the mode superposition method.
- Import Geometry file “beam.stp”

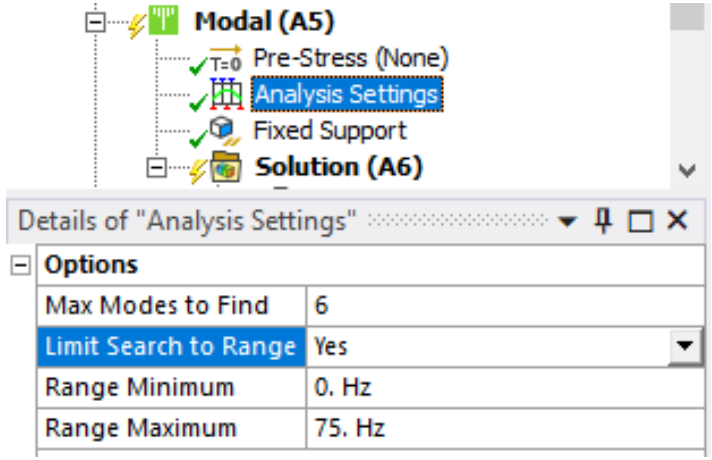


Workshop 06.1 - Preprocessing

- Edit the Model cell to open the Mechanical application.
- In Mechanical, set the units system as follows:
  - Metric (mm, kg, N, s, mV, mA)
  - Degrees
  - RPM
  - Celsius
- For the imported surface body:
  - Assign a Thickness of 25 mm
  - Verify that the material assignment is Structural Steel

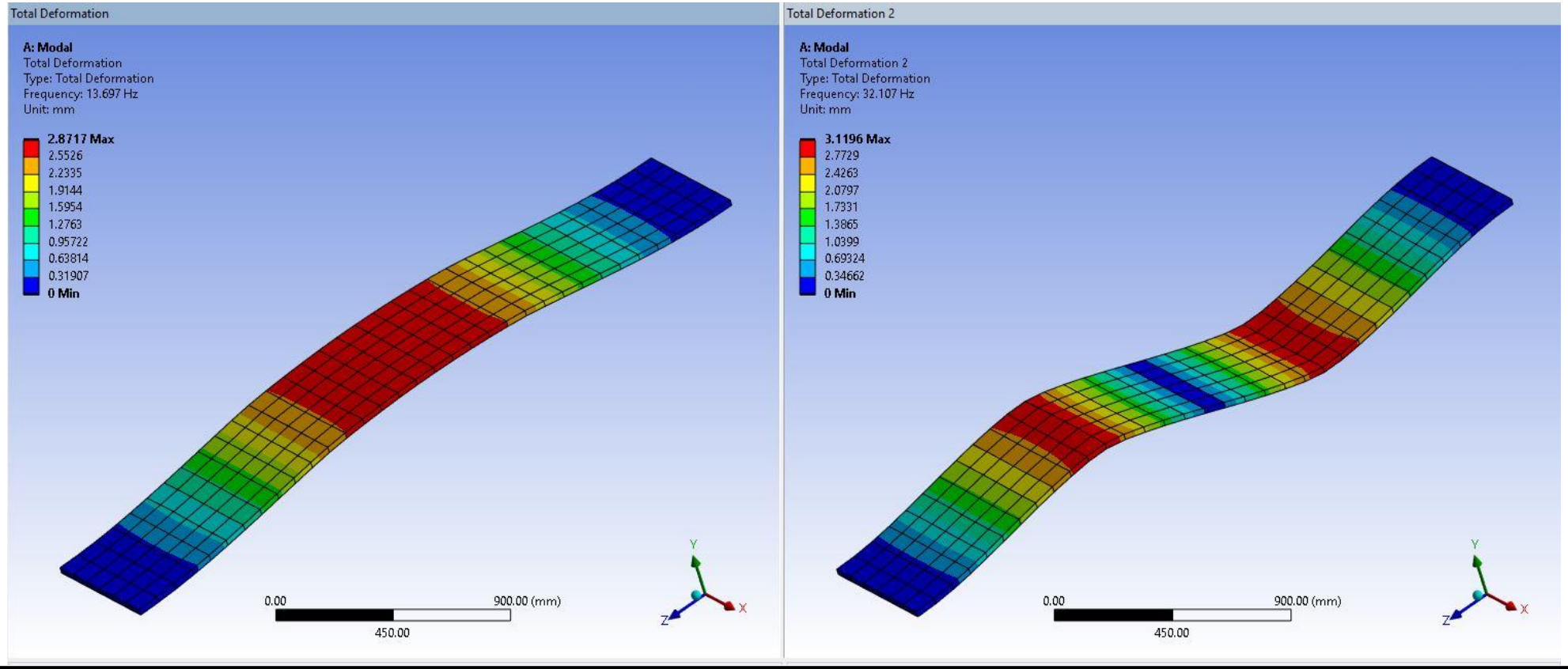
Workshop 06.1 – Modal Environment and Solution Setup

- Constrain both ends of the Beam with Fixed Supports.
- Since the mode shapes and frequencies from the modal analysis will be used in the harmonic response calculation, we want to make sure we have captured sufficient frequency content during the modal analysis. A rule of thumb is to extract modes encompassing 1.5 X the excitation frequency, or modes up to 75 Hz in this example.
  - Use Analysis Settings to Find 6 modes, limiting the search range between 0 and 75 hz.
- Your Settings should look like this:



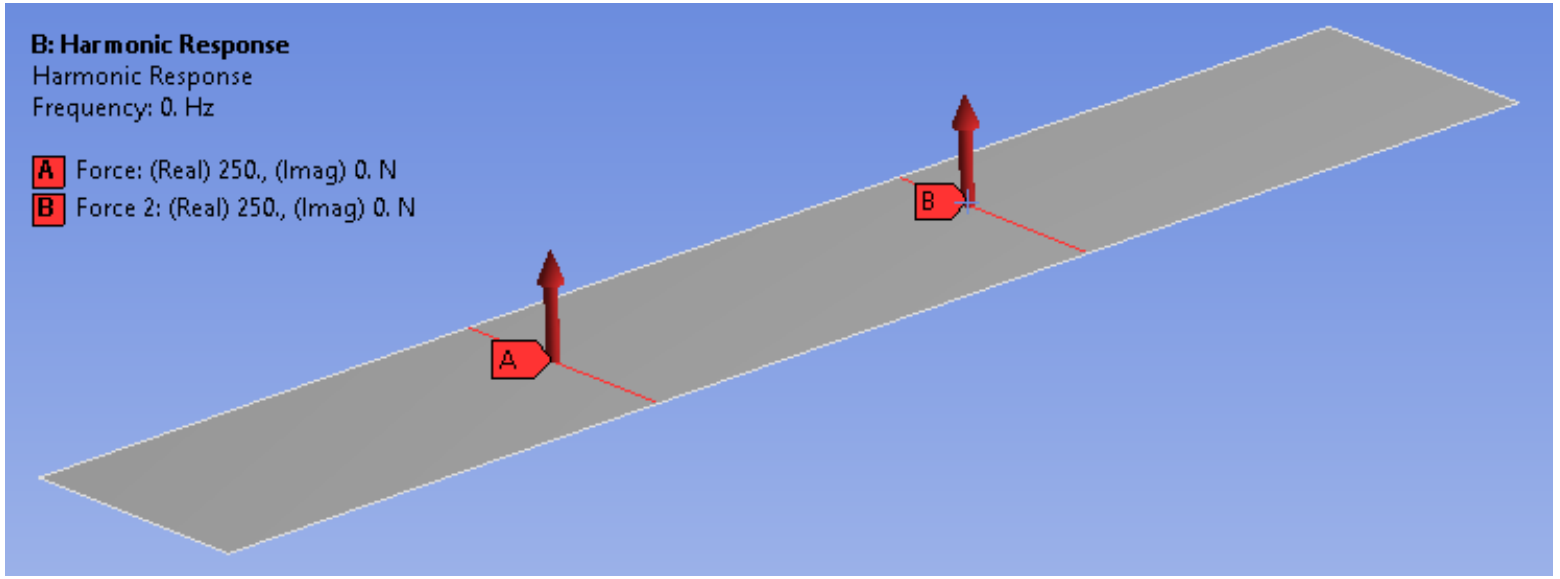
Workshop 06.1 - Modal Results

- Solve the Modal analysis.
- Create some Mode Shape Results to review the results.
  - Note that modes 1 and 2 fall between 0 and 50 Hz



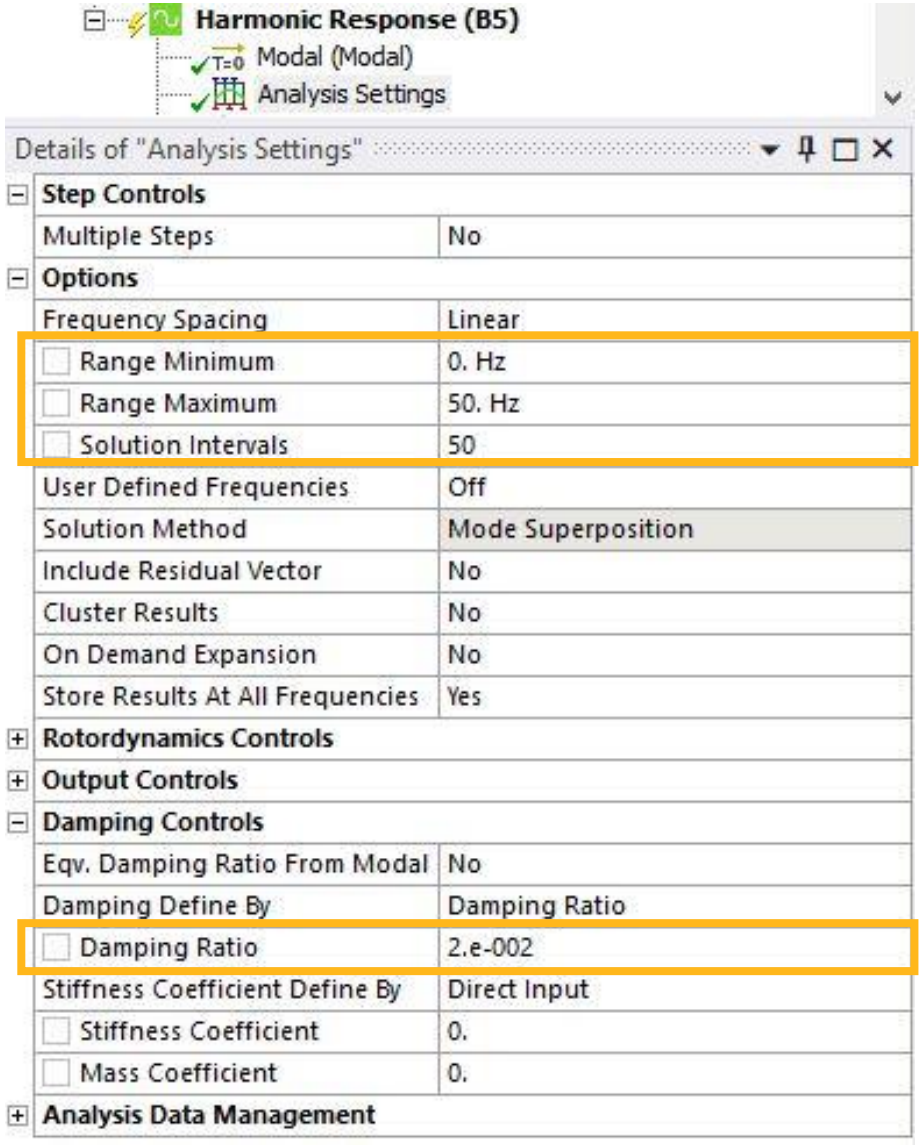
Workshop 06.1 – Harmonic Environment

- There is no need to assign the fixed boundary condition in this Harmonic Response analysis. By transferring the solution from the Modal analysis, this support has been implicitly implemented.
- Apply a 250 N force in the Y direction to each of the two edges on the interior of the surface body.
  - For each force, leave the Phase Angle information set to 0°. We'll investigate differences in phase angles later.



Workshop 06.1 - Harmonic Response Solution

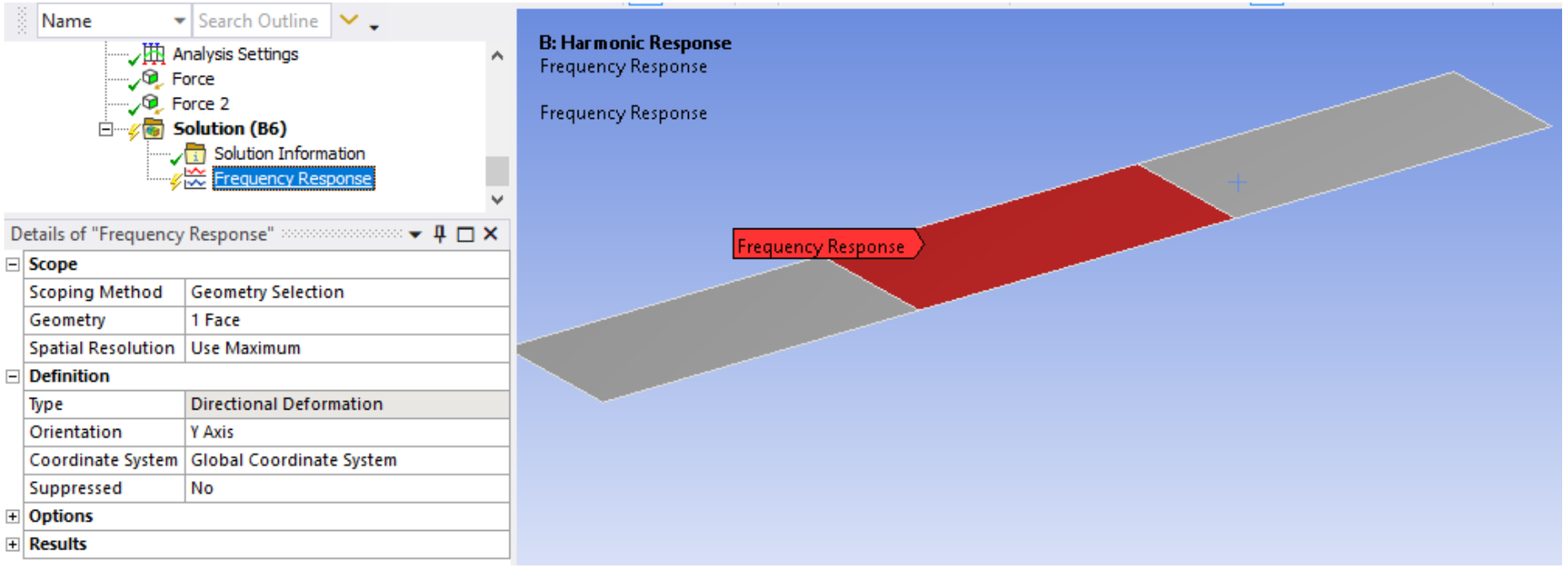
- Edit the Analysis Settings to specify a solution frequency range between 0 and 50 Hz, with solutions spaced evenly at every 1 Hz.
- Without damping, responses at natural frequencies will be unbounded.
  - Define a Damping Ratio of 2%
- Solve the Harmonic analysis.



Workshop 06.1 – Harmonic Results

- Harmonic analyses are generally postprocessed by starting with Frequency Response plots of deformation.
  - Results from the Modal analysis play an important role in determining where we can expect maximum deformation and in which direction it may occur.
  - Review the modal results once again and answer the following questions:
    - To what entities should the frequency response plot(s) be scoped?
    - What direction(s) should we consider?
    - Based upon the answers to these questions, how would you define the frequency response plot(s)? See next slide for a recommended approach.

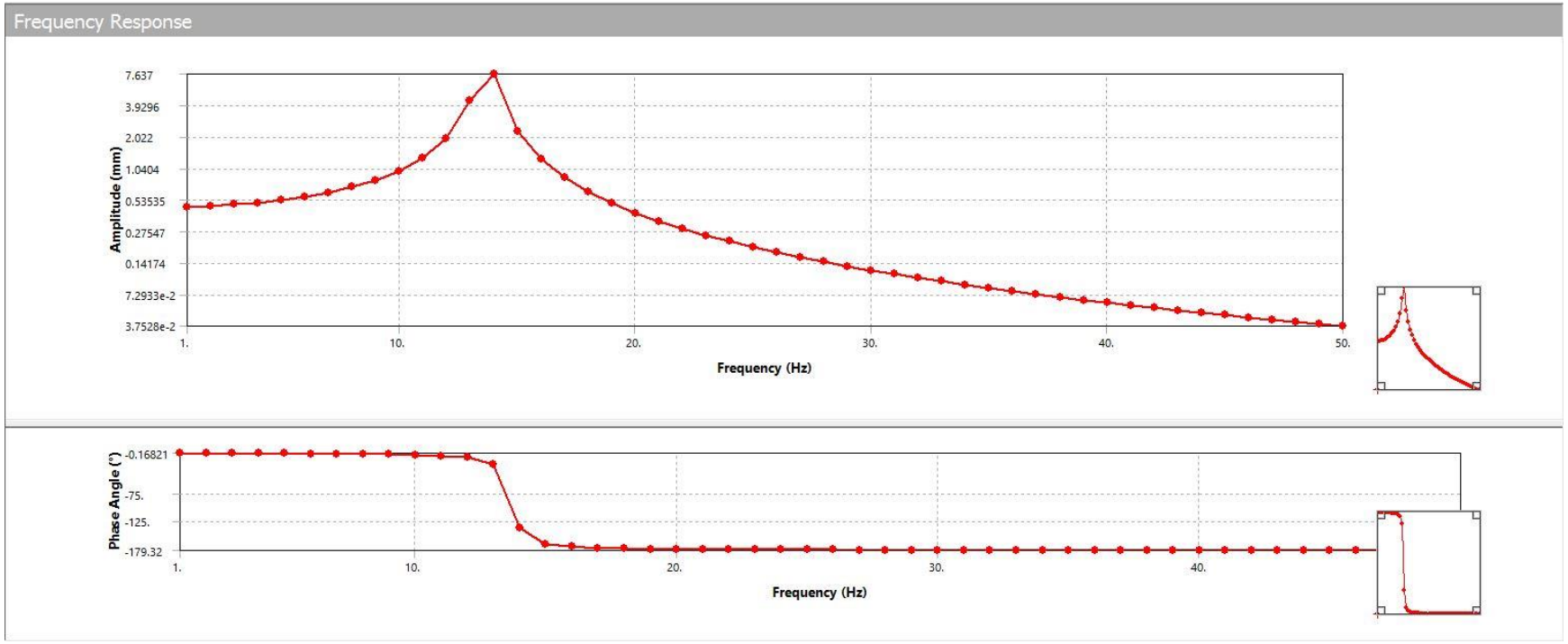
Workshop 06.1 – Harmonic Results



Workshop 06.1 – Harmonic Results

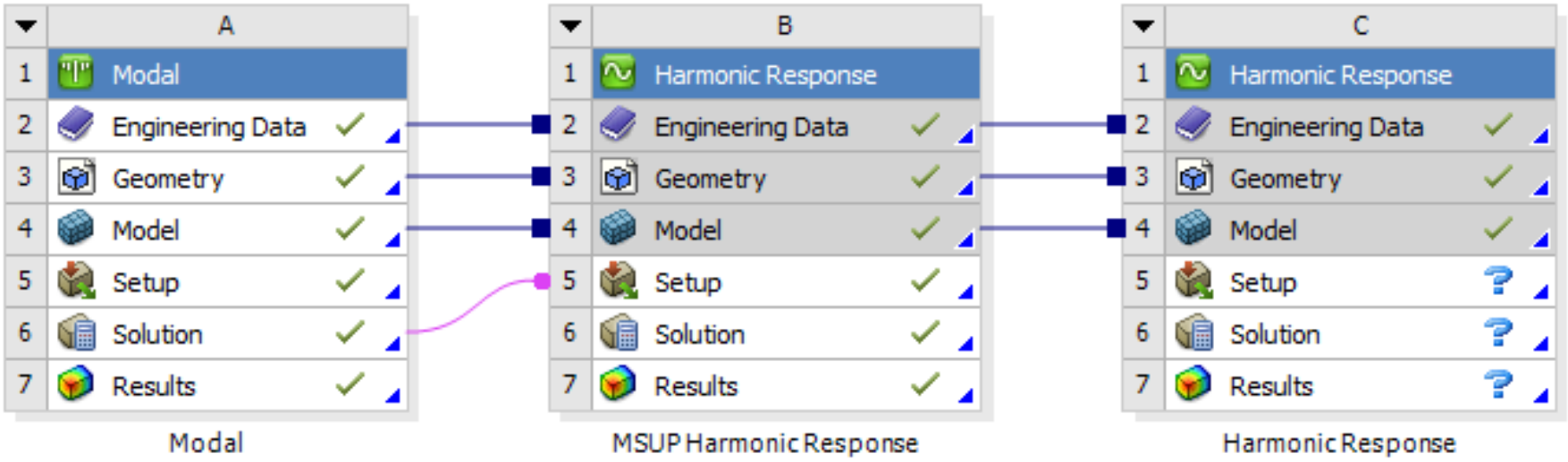
Note: your result magnitudes may vary slightly throughout this workshop due to mesh and software release differences

- Evaluate the results for the frequency response plot.
- Observe the 1st-mode resonance behavior with natural frequency 13.697 Hz and maximum amplitude of 7.637 mm
  - Is this the maximum deformation amplitude? (Recall the first mode frequency, at 13.697 Hz.)
  - Is there reason to believe the response may be larger at 13.697 Hz?



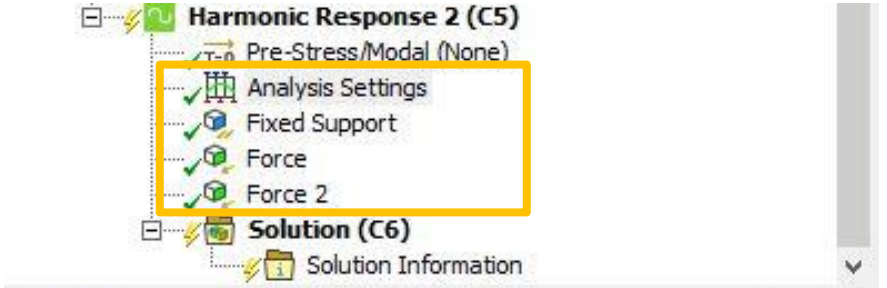
Workshop 06.1 - Mode Superposition vs Full

- An alternative solver for a Harmonic Response analysis system is the Full solver. To use the Full solver, drag and drop a new Harmonic Response system onto the Model cell of the Harmonic system:



Workshop 06.1 - Full solve

- Drag the Fixed Support from the Modal environment and drop it onto the second Harmonic Response environment
- Drag the Forces from the first Harmonic environment and drop them onto the second
- Edit the Analysis Settings:
  - Set solution method to Full
  - Set the Range Minimum to 0 Hz
  - Set the Range Maximum to 50 Hz
  - Set the Solution Intervals to 150 (resulting in solutions at 0.333 Hz intervals)
  - Set the Structural Damping Coefficient to .04*
- Solve the Harmonic analysis.



Details of "Analysis Settings"

Step Controls	
Options	
Frequency Spacing	Linear
Range Minimum	0. Hz
Range Maximum	50. Hz
Solution Intervals	150
User Defined Frequencies	Off
Solution Method	Full
Variational Technology	Program Controlled
Rotordynamics Controls	
Advanced	
Output Controls	
Damping Controls	
Constant Structural Damping Coefficient	4.e-002
Stiffness Coefficient Define By	Direct Input
Stiffness Coefficient	0.
Mass Coefficient	0.
Analysis Data Management	

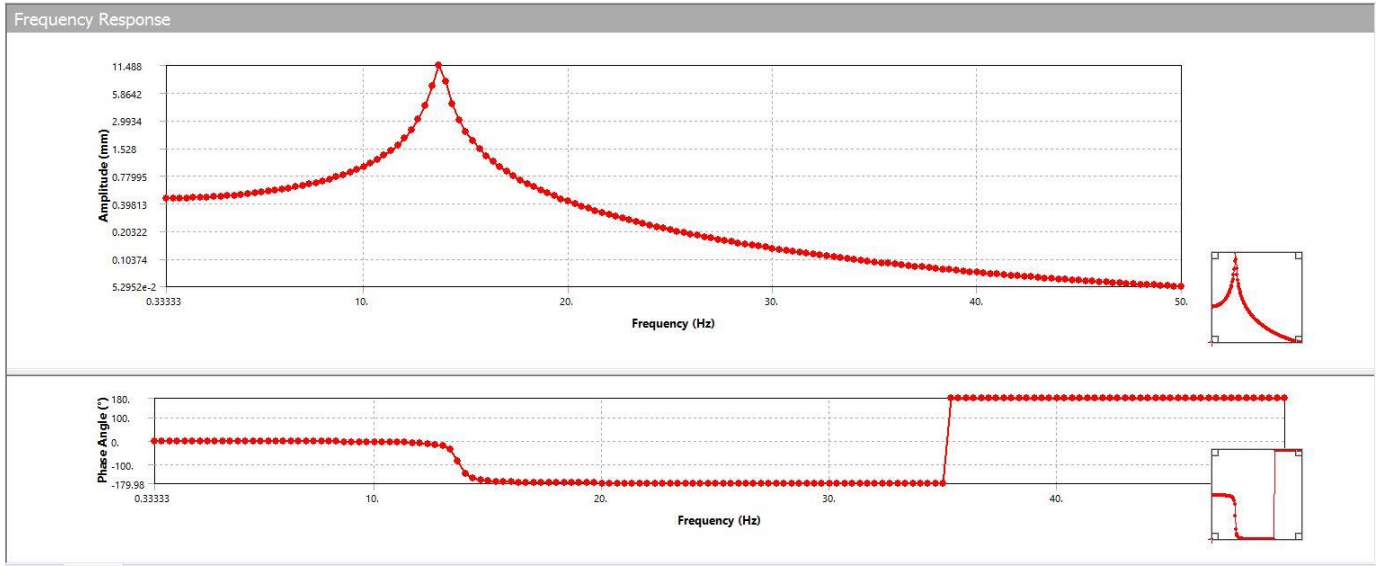
Damping Coefficient enters the damping contribution as g[K], whereas Damping Ratio enters it as 2g[K]; therefore, we use 2.02 = .04 in the Full Method to achieve a direct comparison with results from MSUP.

Workshop 06.1 - Full method results

Details of "Frequency Response"

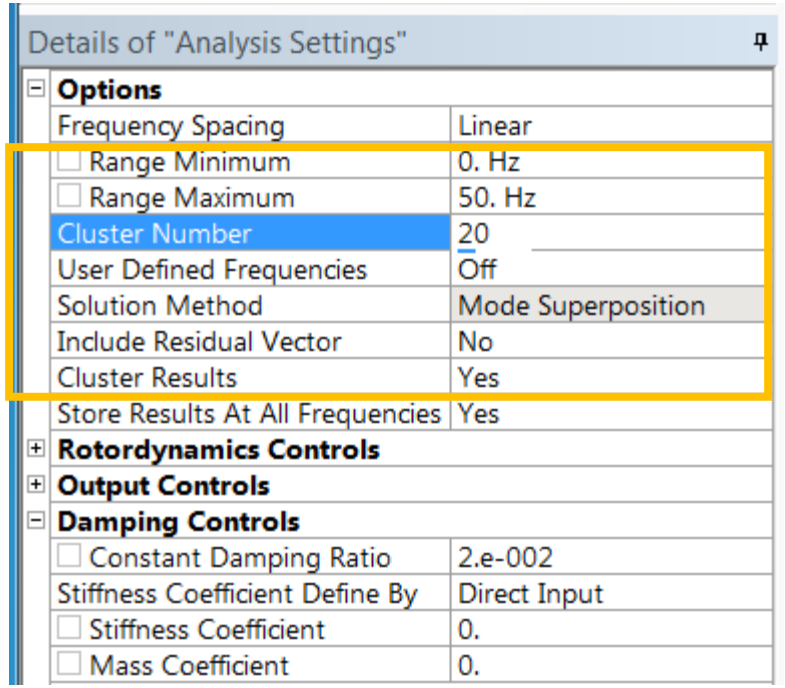
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Spatial Resolution	Use Maximum
Definition	
Type	Directional Deformation
Orientation	Y Axis
Coordinate System	Global Coordinate System
Suppressed	No
Options	
Frequency Range	Use Parent
Minimum Frequency	0. Hz
Maximum Frequency	50. Hz
Display	Bode
Chart Viewing Style	Log Y

- Insert a Deformation Frequency Response.
  - Set the scoping to the center face of the beam
  - Set the Spatial Resolution to Use Maximum
  - Set the Orientation to Y Axis
- Observe the 1st-mode resonance behavior with natural frequency 13.667 Hz and maximum amplitude of 11.488 mm
  - Comparing this with the amplitude at 15 Hz obtained from the mode superposition analysis, we see a significant difference; we can expect large changes in response at frequencies at/near resonance.



Workshop 06.1 – MSUP with clustering

- Can we improve the accuracy of the mode-superposition results?
  - Clustering helps to increase the accuracy of the solution by clustering the frequency intervals around the natural frequencies, rather than using equally spaced frequency intervals.
- Return to the Harmonic Response system that used the Mode Superposition method. Switch Cluster Results to Yes, and change Cluster Number to 20



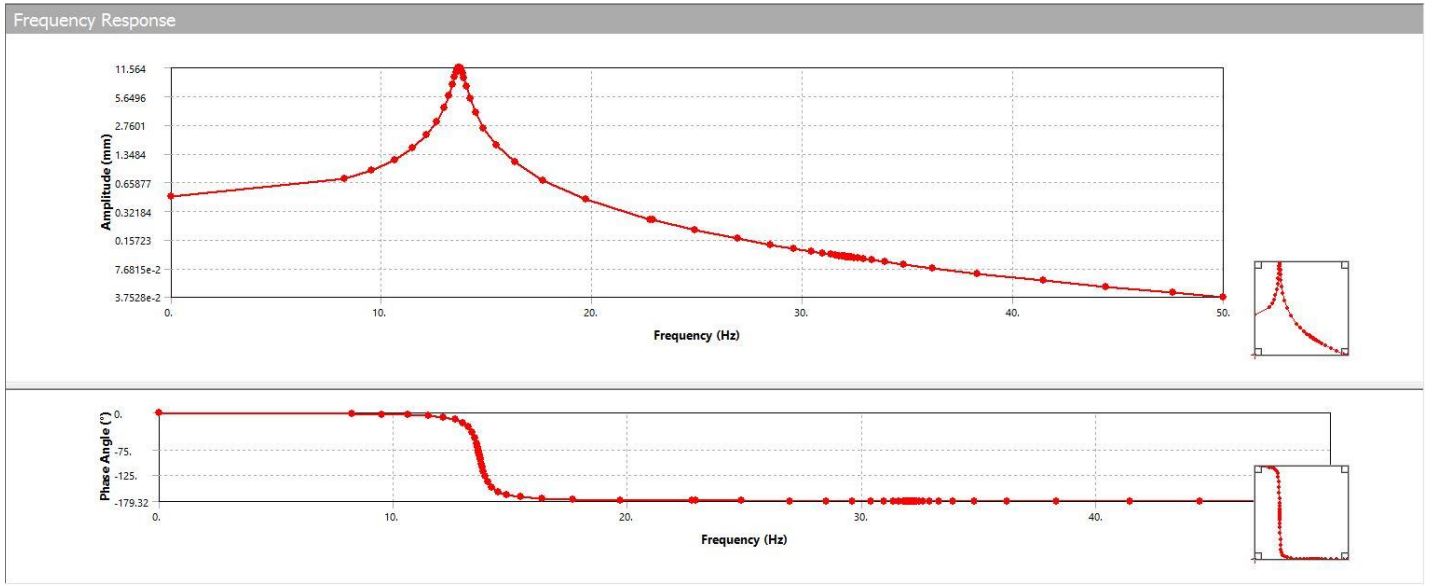
Workshop 06.1 - MSUP with clustering

- Re-solve the MSUP Harmonic Response.
- Insert a Deformation Frequency Response.
  - Set the scoping to the center face on the beam
  - Set the Spatial Resolution to Use Maximum
  - Set the Orientation to Y Axis

Details of "Frequency Response"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Spatial Resolution	Use Maximum
Definition	
Type	Directional Deformation
Orientation	Y Axis
Coordinate System	Global Coordinate System
Suppressed	No
Options	
Frequency Range	Use Parent
Minimum Frequency	0. Hz
Maximum Frequency	50. Hz
Display	Bode
Chart Viewing Style	Log Y

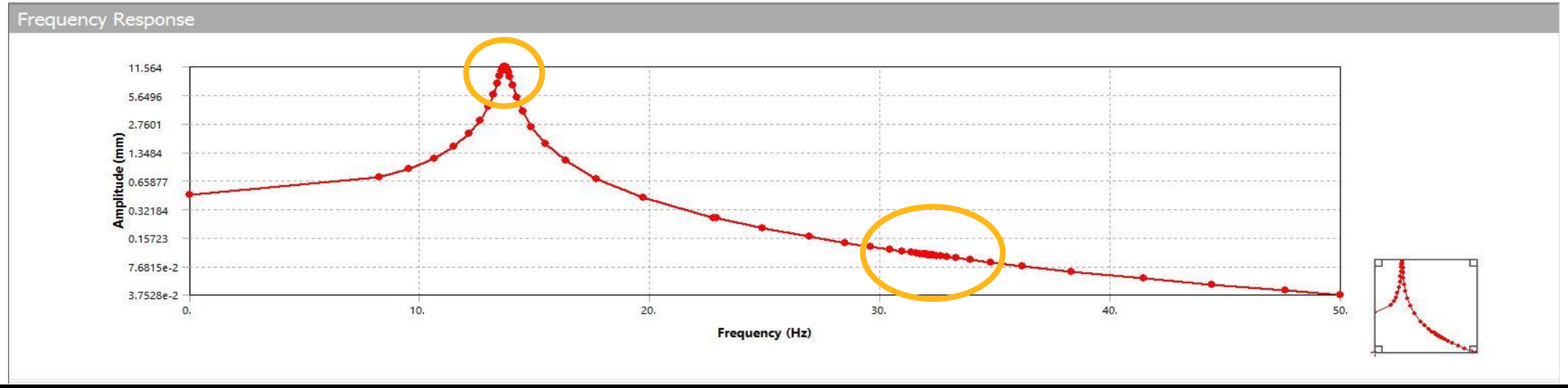
- Observe the 1st-mode resonance behavior with natural frequency 13.697 Hz and maximum amplitude of 11.564 mm



Workshop 06.1 - Solver comparison

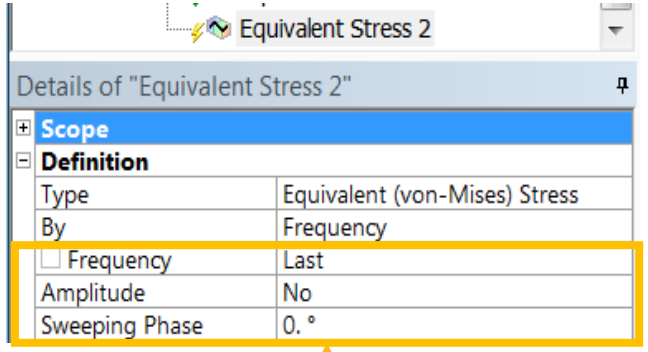
MSUP (50 intervals)	Full (50 intervals)	MSUP with clustering (20)
7.637mm	11.488mm	11.564mm

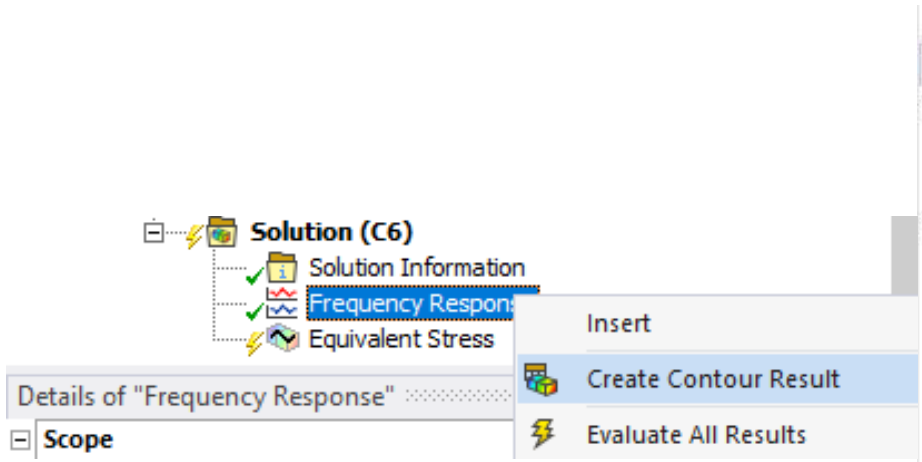
- As we can see, the MSUP and Full results did not compare favourably at first. In theory, the MSUP would not be as accurate as the Full method, as it relies on the superposition of the natural modes to describe the harmonic response.
- However, MSUP with the clustering option gives additional accuracy to the MSUP method as it clusters the analysis frequencies around the natural frequencies. The chart below shows clustering around the 1st and 2nd natural frequencies.

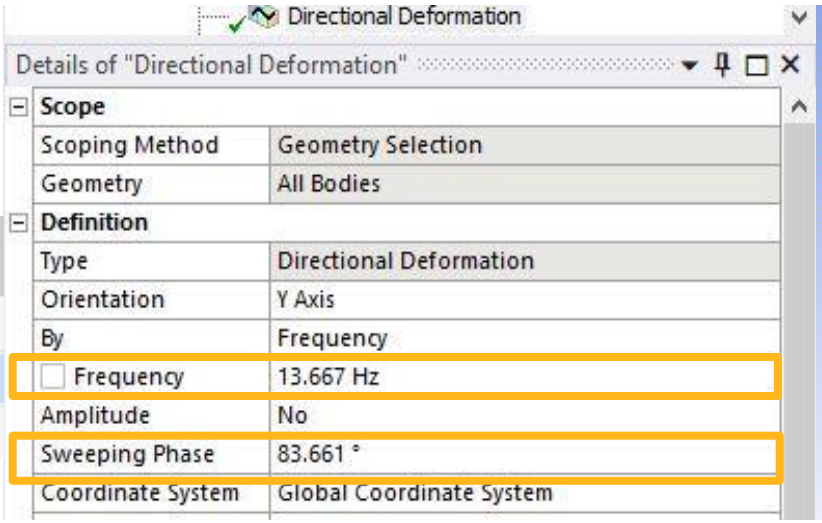


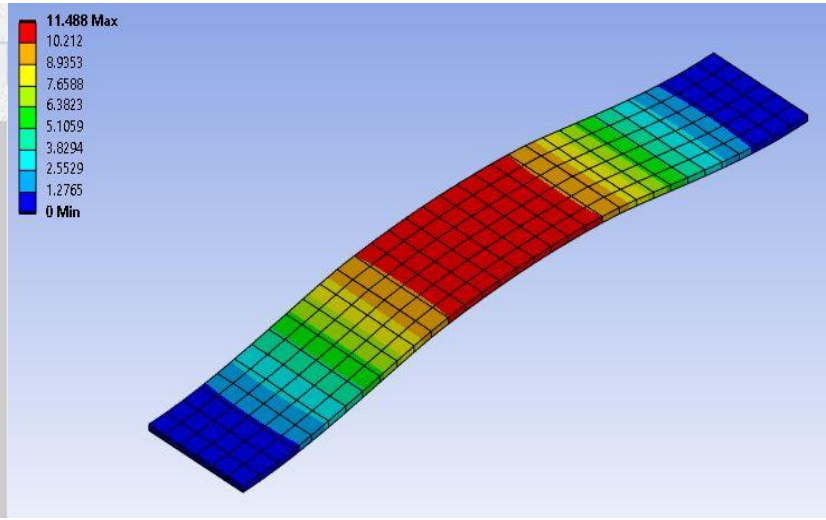
Workshop 06.1 - Results

- You can also plot contours at specific frequencies.
- Click RMB on the solution object and > Insert> Stress, Strain, or Deformation
  - This will insert the result object(s)
  - It is necessary to specify a specific frequency and sweeping phase.
- Or, to ensure we obtain the peak contour at the peak displacement amplitude and corresponding phase, RMB on the Frequency Response and choose “Create Contour Result”



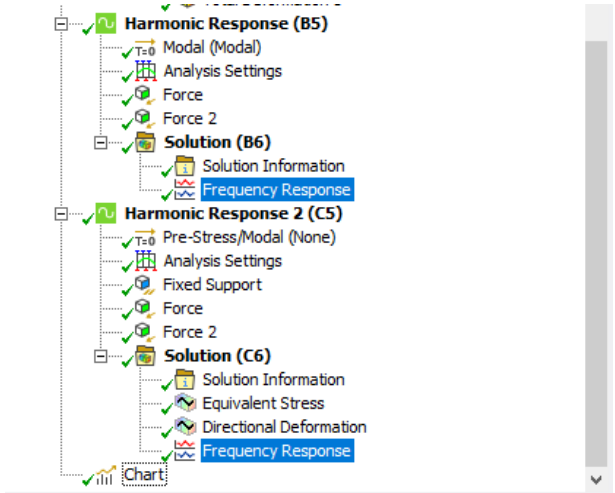


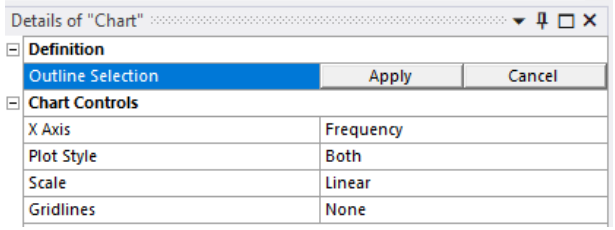


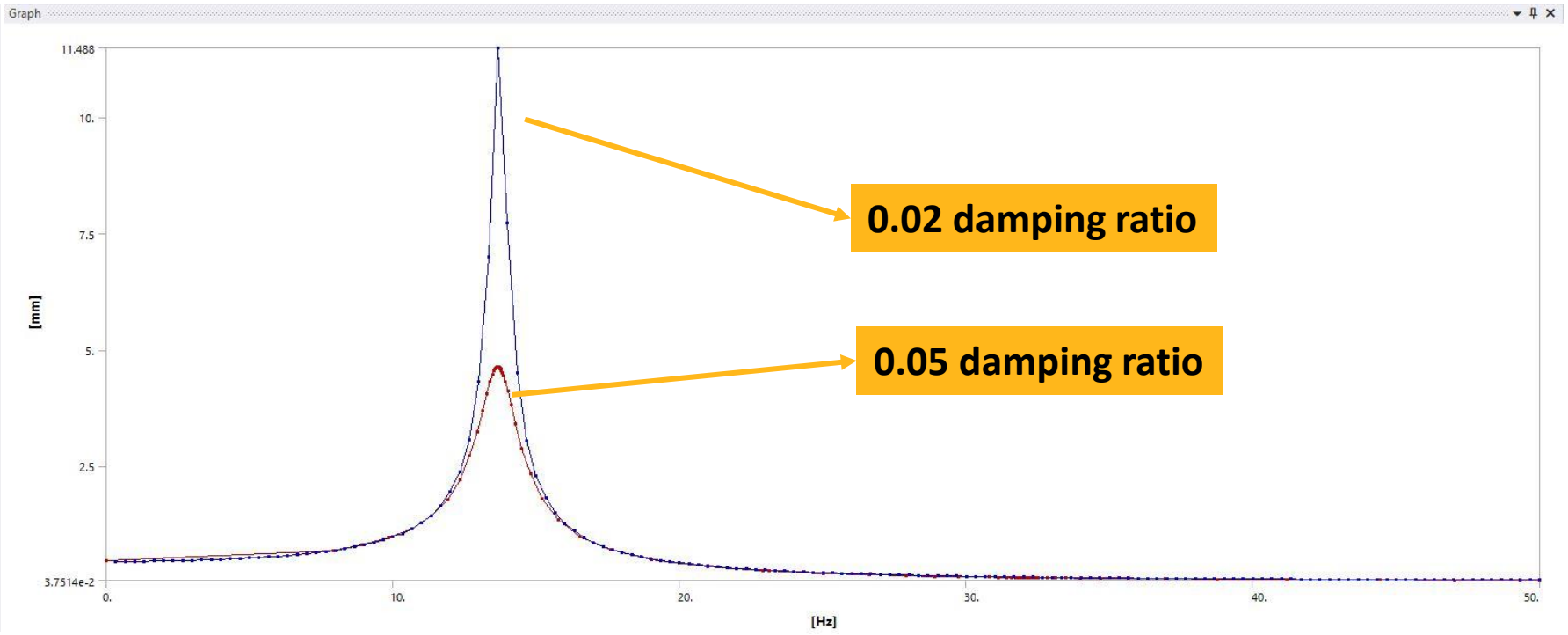


Workshop 06.1 - Results (Damping)

- Return to Analysis Settings of the MSUP model with clustering.
  - Adjust the damping ratio from 0.02 to 0.05
- Solve and compare with the frequency response plot from the full solution method by creating a Chart object.
  - It can be seen that damping properties have a significant effect on the dynamic behavior of the structure around the resonant frequencies.

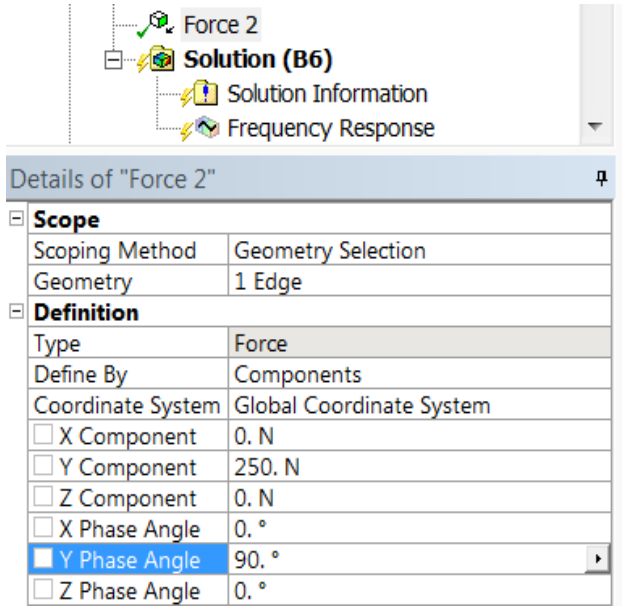


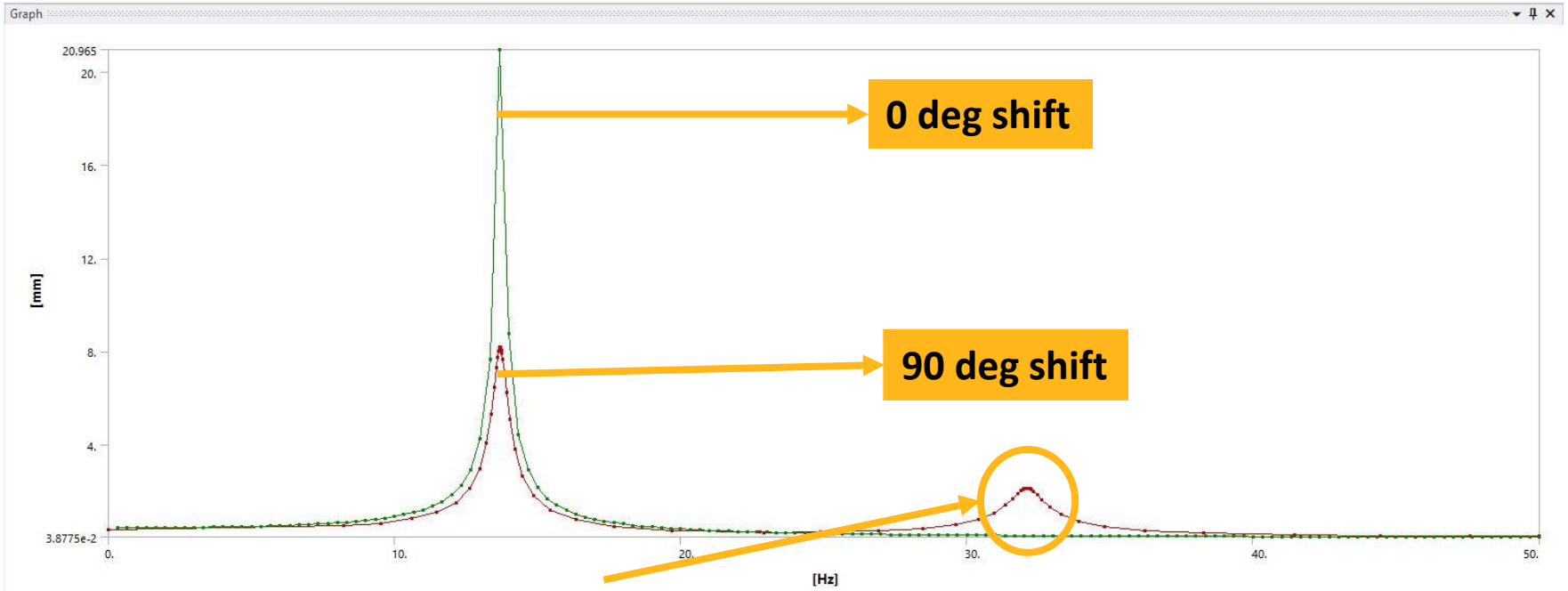




Workshop 06.1 - Results

- Return to Analysis settings and change the damping back to 0.02
- Under the 2nd Force applied,
  - set the Phase Angle to 90° (we will try to excite different modes)
  - resolve the Harmonic Response





- Observe that the load phase shift has changed the first resonance behavior significantly, and has also excited the 2nd natural frequency at approximately 40 Hz.

End of presentation