Ansys Mechanical Linear and Nonlinear Dynamics

WS 09.2: Gantry Crane

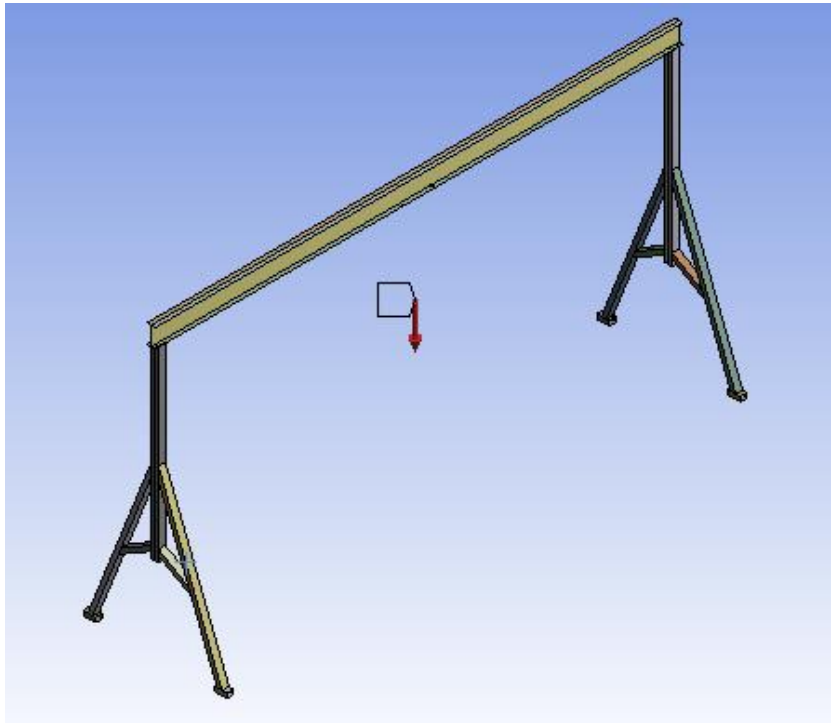
Release 2022 R2

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.

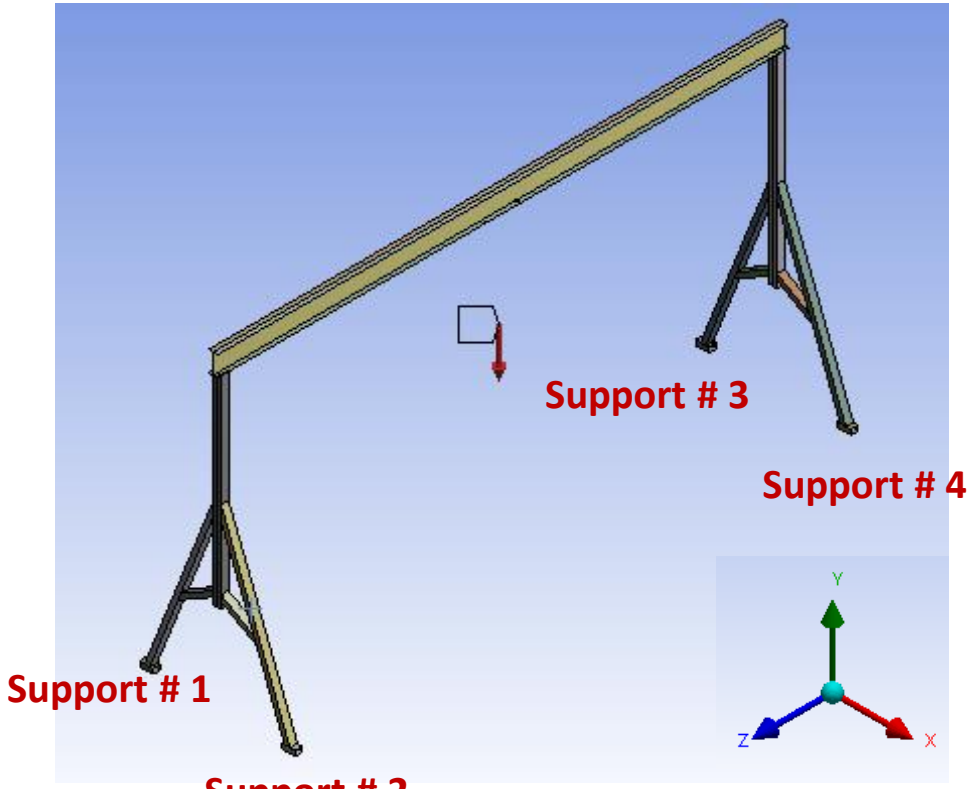
Workshop 09.2 - Goals

- This workshop consists of a gantry crane assembly. The crane is rated to carry a 50,000 N weight applied as an impulse over 0.1 seconds.
- The purpose of this workshop is to compare the full transient solution method to the mode superposition solution method.



Workshop 09.2 - Assumptions

- Supports 1 through 3 are fixed in all directions, Support 4 is free to move in the X-direction.
- A remote force of 50,000 N is applied and is scoped to the underside of the horizontal beam, midway between the supports.

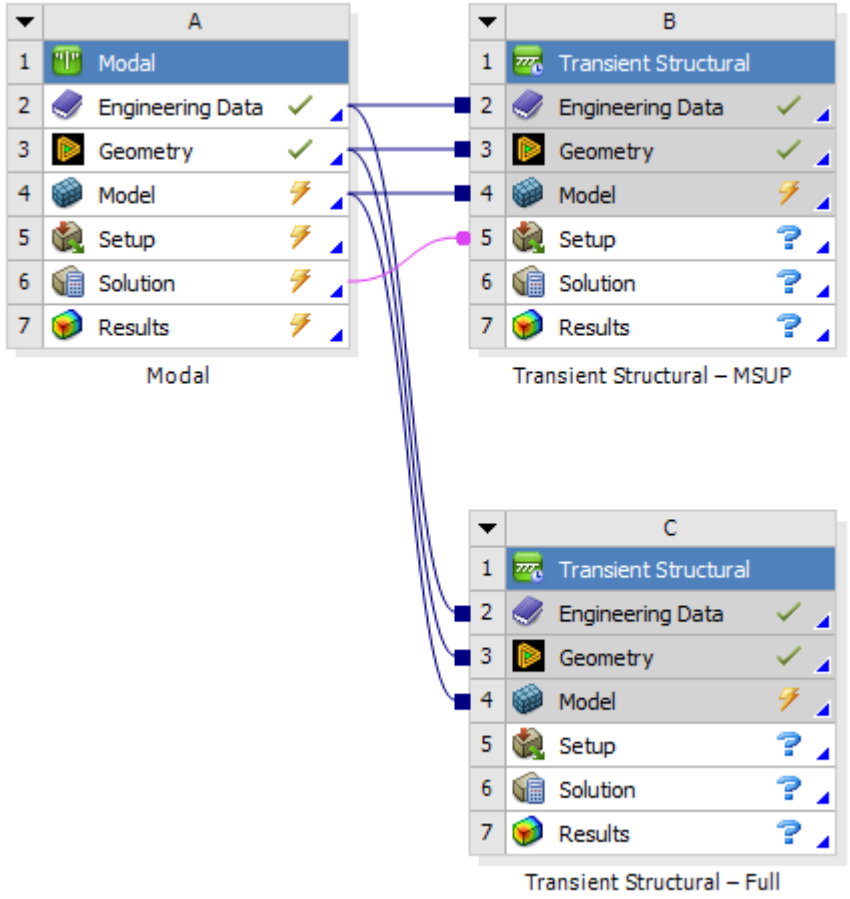


Workshop 09.2 - Project Schematic

- Start a new Workbench session and open the archive file “WS09.2-Gantry_Crane.wbpz” supplied with this workshop.
- The archive consists of a Modal analysis from which the Mode Superposition (MSUP) will be conducted.
- Drag a Transient Structural solution on to the Solution cell of the Modal analysis in order to set up the MSUP solution.
- Drag a second Transient Structural solution on to the Model cell of the Modal analysis in order to set up the full transient solution.
- Rename each Transient Structural system to reflect the solution method being used, for example “Transient Structural – MSUP” and “Transient Structural – Full”

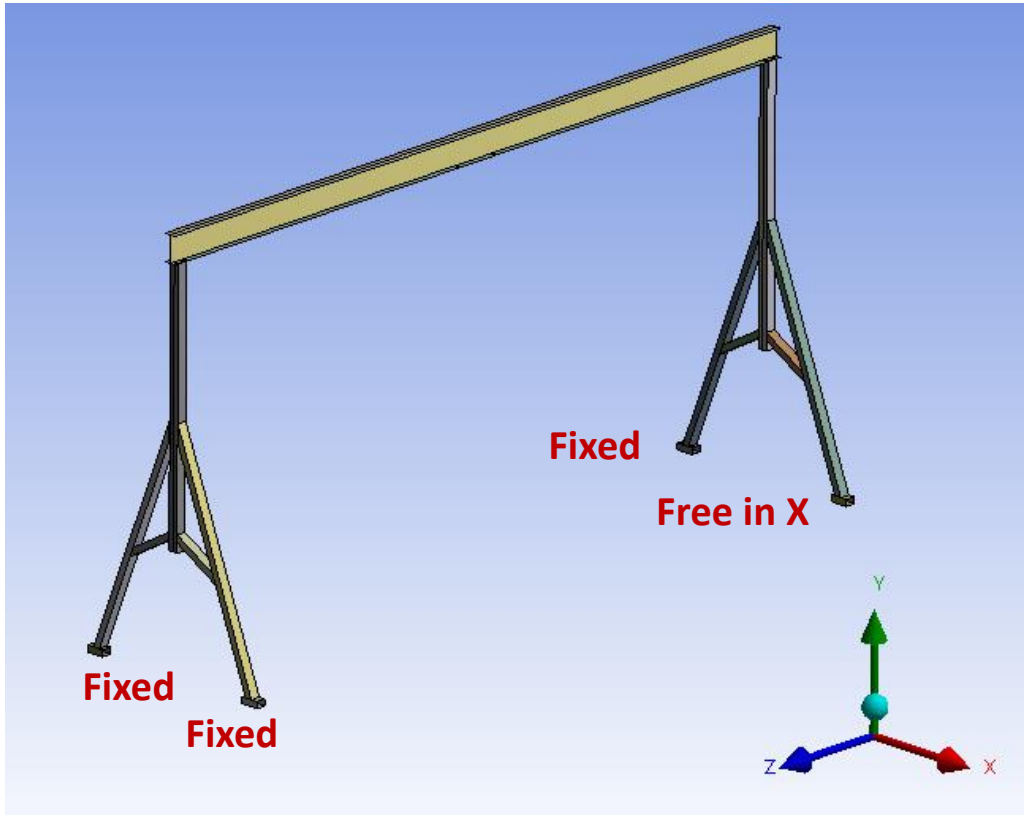
Workshop 09.2 - Project Schematic

- The Project Schematic should look like this when complete.



Workshop 09.2 – Modal Analysis Setup

- Open Mechanical from the Modal analysis system.
- This workshop will be solved in Metric units: “Metric (m, kg, N, s, V, A)”.
- Apply the appropriate supports to enforce the conditions depicted below.



Workshop 09.2 – Modal Analysis Solution

- Solve the modal analysis using the default of 6 modes to extract.
- Review the mode shapes for each mode, along with the Participation Factor Summary.
  - What mode(s) do you expect to be excited by the impulse load on the crane?

Tabular Data

	Mode	Frequency [Hz]
1	1.	3.4135
2	2.	4.4385
3	3.	8.5622
4	4.	9.7807
5	5.	13.573
6	6.	14.575

Click Here to see a demonstration of the model setup to this point

Participation Factor

Mode	Frequency [Hz]	X Direction	Y Direction	Z Direction	Rotation X	Rotation Y	Rotation Z
1	3.4135	18.263	-0.12806	-9.5808e-002	-1.1272	-81.3	-49.983
2	4.4385	5.787e-003	4.357e-002	24.441	53.418	17.763	-0.2177
3	8.5622	-0.95684	0.72103	-7.0652e-002	4.7515	54.961	-0.45581
4	9.7807	0.80199	-6.0888e-002	3.1417e-003	-0.38909	-3.4929	0.21724
5	13.573	-9.4793	4.631	-7.4149e-002	24.56	49.471	17.563
6	14.575	0.34433	16.773	-4.1835e-002	76.553	4.2313	-17.915

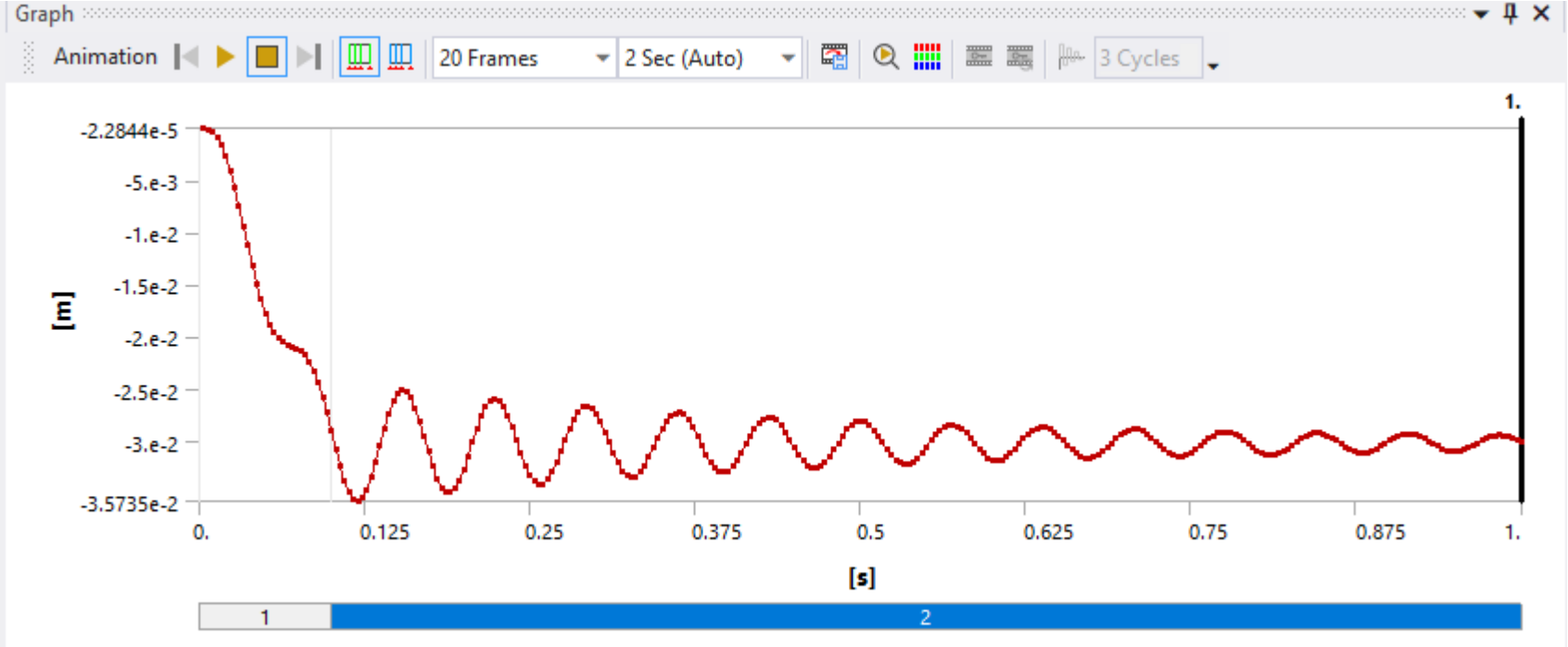
Note: your result magnitudes may vary slightly throughout this workshop due to mesh and software release differences

Workshop 09.2 – MSUP Transient Setup

- Use a 2-step solution to solve the MSUP Transient analysis.
  - The first step will span a duration from 0.0 to 0.1 seconds.
  - The second step will span a duration from 0.1 seconds to 1 second.
  - Use 30 solution substeps during step 1, and 270 substeps during step 2.
- Damping
  - Define a stiffness-based damping coefficient based upon a 2.5% damping ratio and the expected response frequency obtained from the modal analysis.
- Apply the impulse force as a remote force to the two surfaces on the underside of the horizontal beam, ramping it from 0 to 50,000 N (-Y direction) during the first step. Change the Y coordinate of the force to 1.1 m.
- Keep the impulse force constant at 50,000 N during the second step.
- Are supports needed during the MSUP Transient solution?

Workshop 09.2 – MSUP Transient Solution and Results

- Solve the MSUP Transient model.
- Examine the directional deformation of the entire structure in the Y direction.
  - Does the response frequency agree with the results from the modal analysis?



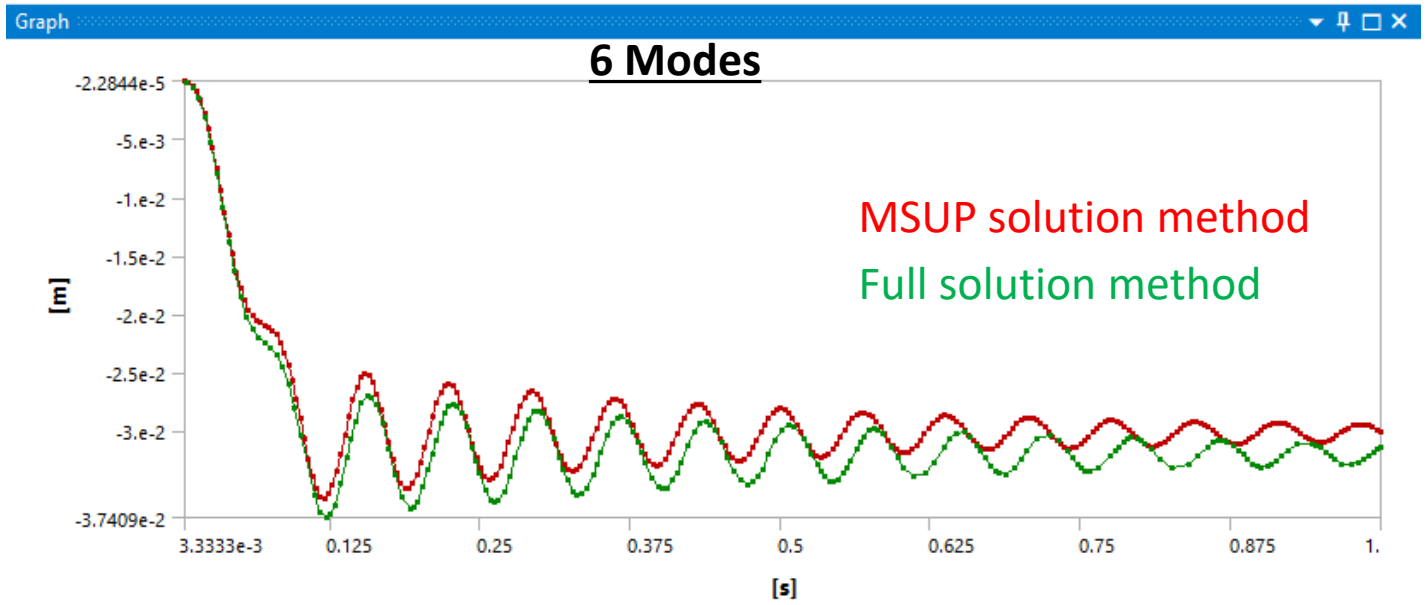
Click Here to see a demonstration of the MSUP transient solution

Workshop 09.2 – Full Transient Setup

- Use a 2-step solution for the full transient analysis with end times for each step identical to the MSUP analysis.
  - Use 20 initial substeps for steps 1 and 2
  - Use 20 minimum substeps for steps 1 and 2
  - Use 50 and 1000 maximum substeps for steps 1 and 2 respectively
- Apply the same damping in the full transient setup.
- Copy the supports and applied force from the other environments into the full transient environment.
- Copy the Directional Deformation result from the MSUP transient into the full transient solution folder.

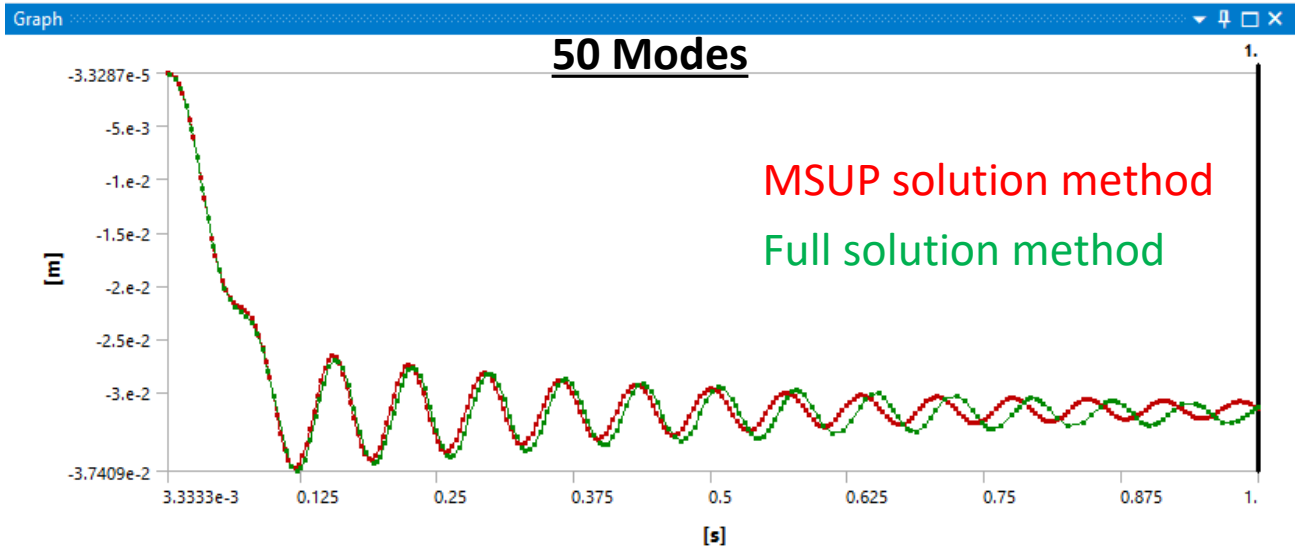
Workshop 09.2 – Full Transient Solution and Results

- Solve the full transient analysis (it may take approx. 20 minutes).
- Review the Directional Deformation result.
  - How well does it compare to the MSUP deformation result?
- Compare the two plots by creating a Chart object in Solution, selecting the total deformation results from each of the solution methods.



Workshop 09.2 – Full Transient Solution and Results

- Why are the two solutions different?
  - What is the ratio of effective mass to total mass in the Y direction represented by the modal analysis?
  - Is each solution method linear or nonlinear?
- Attempt to achieve a better correlation between the two solutions by increasing the number of modes extracted to 50 in the modal analysis.
  - What's the ratio of effective mass to total mass now?



Click Here to see a demonstration of the Full transient and refined solutions

End of presentation