Ansys Mechanical Linear and Nonlinear Dynamics

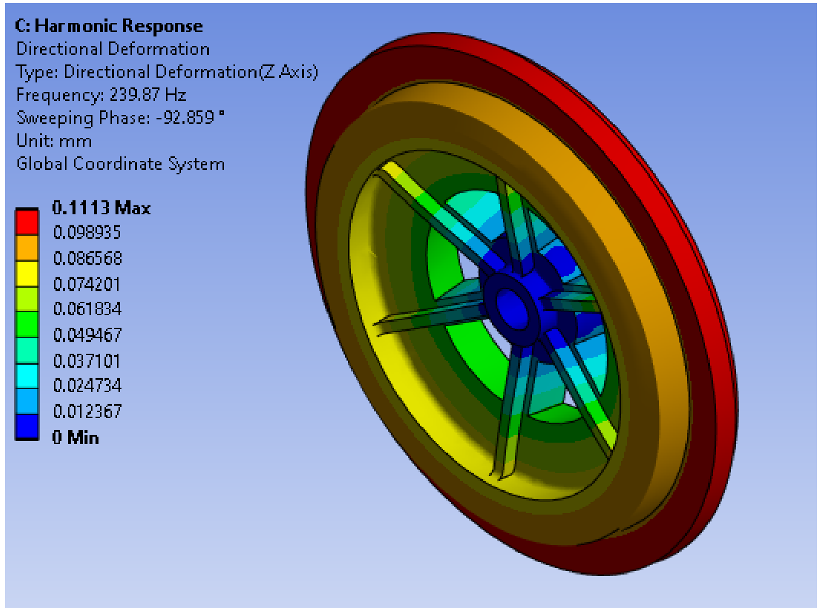
WS 01.1: Flywheel

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 01.1 – Introduction

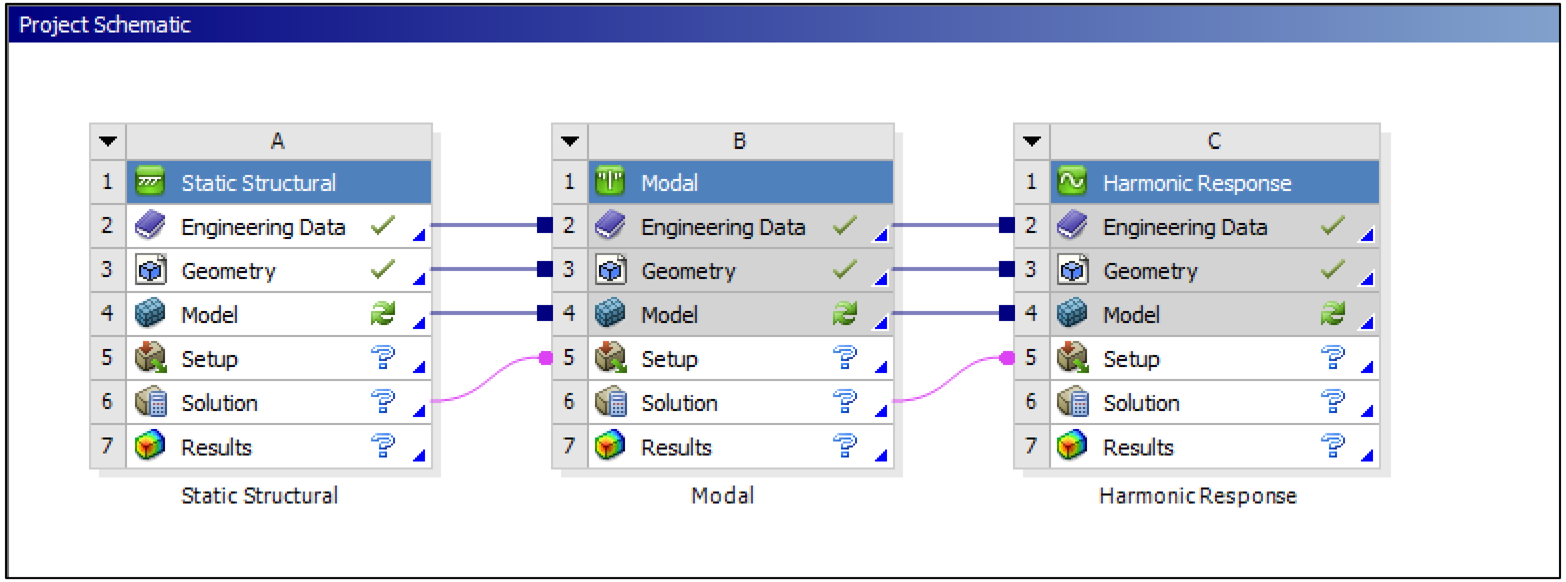
- In this workshop, the vibration characteristics of a spinning flywheel will be investigated, along with its response to a harmonic load.
- Spinning structures and those under high-tension loading often exhibit natural frequencies different than those that might exist for the same structure in an un-loaded state.
    - This can be accounted for by performing a pre-stressed modal analysis, followed by a mode-superposition harmonic analysis, in which mode shape results are used to calculate harmonic response.



Workshop 01.1 – Project Schematic

- Drop a Static Structural system into the Project Schematic.
  - In this system, a 600 RPM rotational velocity will be applied to pre-stress the flywheel.
- Drop a Modal Analysis on to the Solution cell of Static Structural system.
  - This will perform the pre-stressed modal analysis.
- Drop a Harmonic Analysis on to the Solution cell of the Modal system.
  - This will perform the mode-superposition harmonic analysis using an applied harmonic acceleration of approximately 2G's in the Z direction over a frequency range from 0 – 500 Hz.
- The geometry for this workshop is provided in an iges file format
  - Import "**Flywheel.igs**" in the Geometry cell of the Static Structural system.
- The project schematic should appear as shown on the next slide.

Workshop 01.1 – Project Schematic



Workshop 01.1 – Units

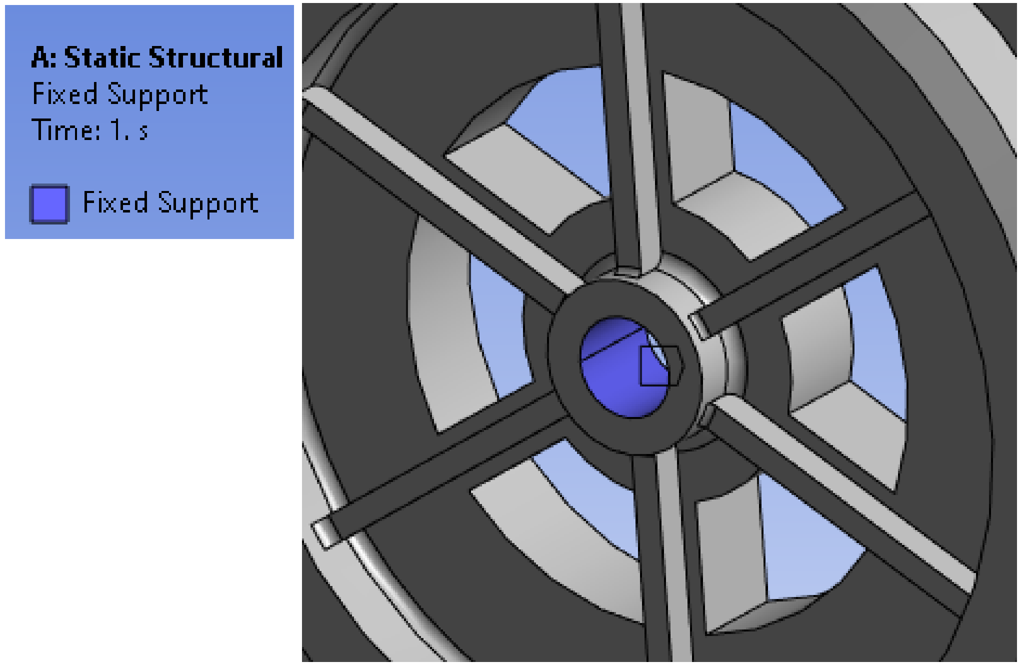
- Open the Static Structural model In Mechanical, set the units system as follows:
  - Metric (mm, kg, N, s, mV, mA)
  - Degrees
  - RPM
  - Celsius

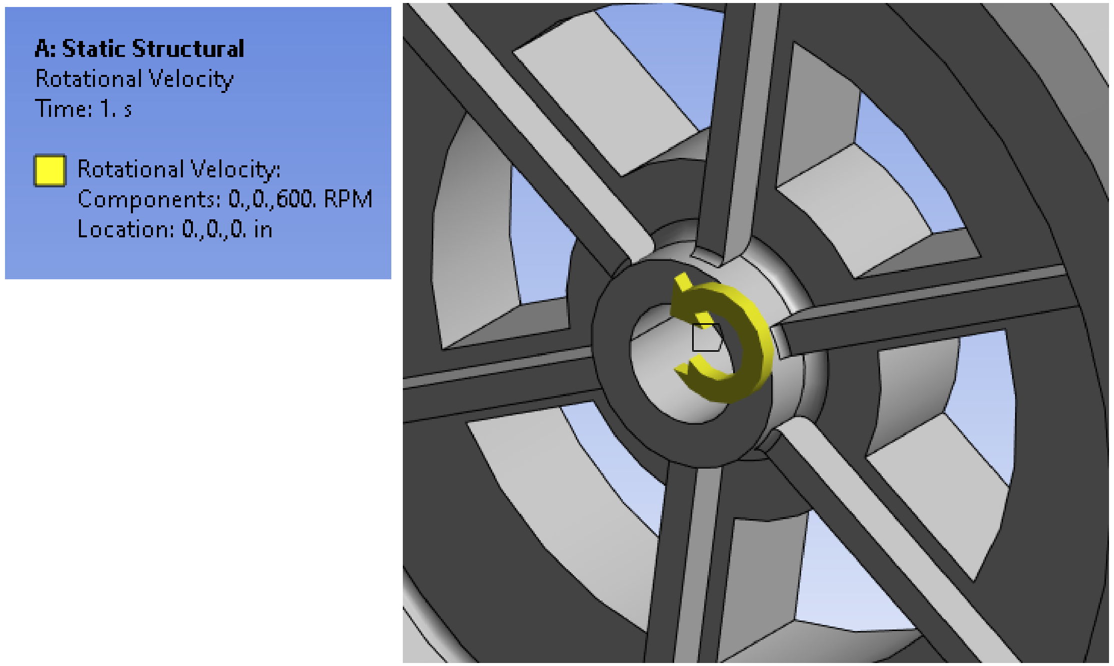
Click Here to see a demonstration of setting up the project

Workshop 01.1 – Static Preprocessing

- We'll simulate the effects of the flywheel fixed to a shaft spinning at 600 RPM.
  - Apply a fixed support at the center hole of the flywheel.
  - Apply a rotational velocity inertial load of 600 RPM about the Z axis.
- Solve the static structural model.

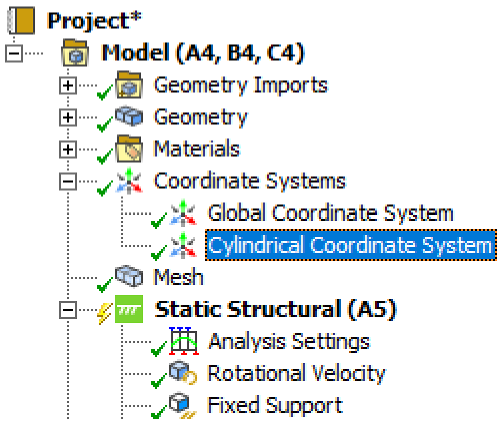
Click Here to see a demonstration of setting up the Static analysis

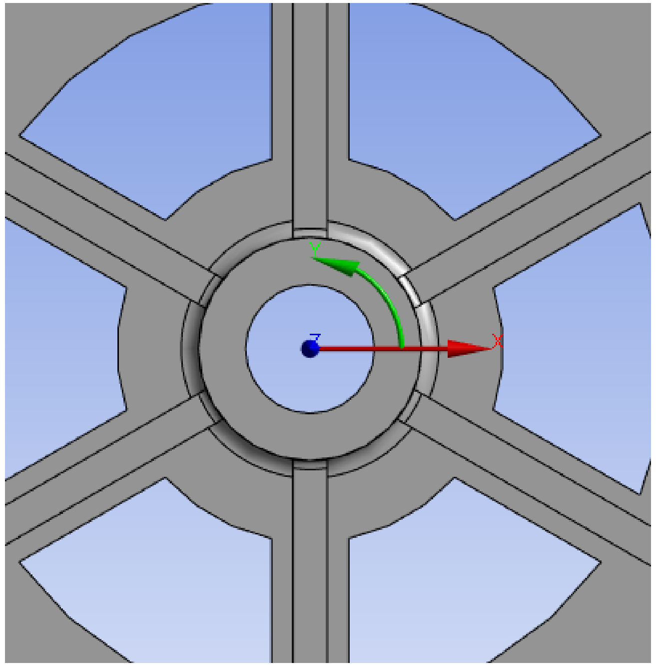




Workshop 01.1 – Postprocessing

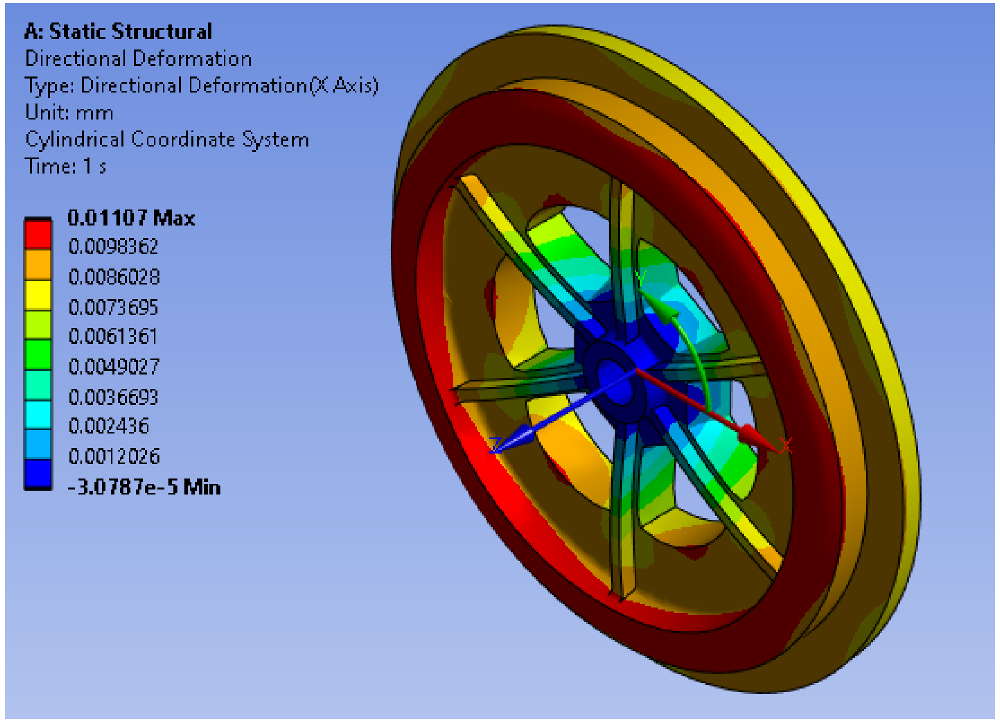
- *Note: your result magnitudes may vary slightly throughout this workshop due to mesh and software release differences.*
- We'll postprocess the radial and tangential deformation/stresses in the flywheel.
  - Create a new cylindrical coordinate system at the global origin, aligned with Global Cartesian.
  - You may want to rename the system for easy identification.

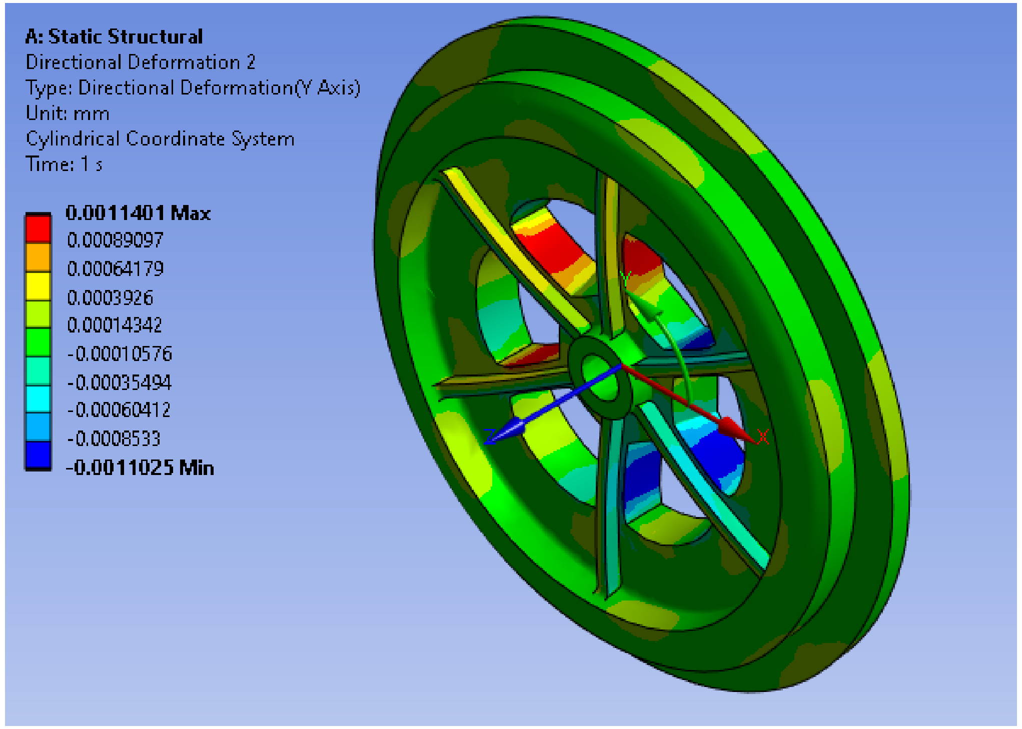




Workshop 01.1 – Static Postprocessing

- Insert Directional Deformation results in the X and Y directions, referencing the newly created Cylindrical Coordinate System.
- Evaluate these results.

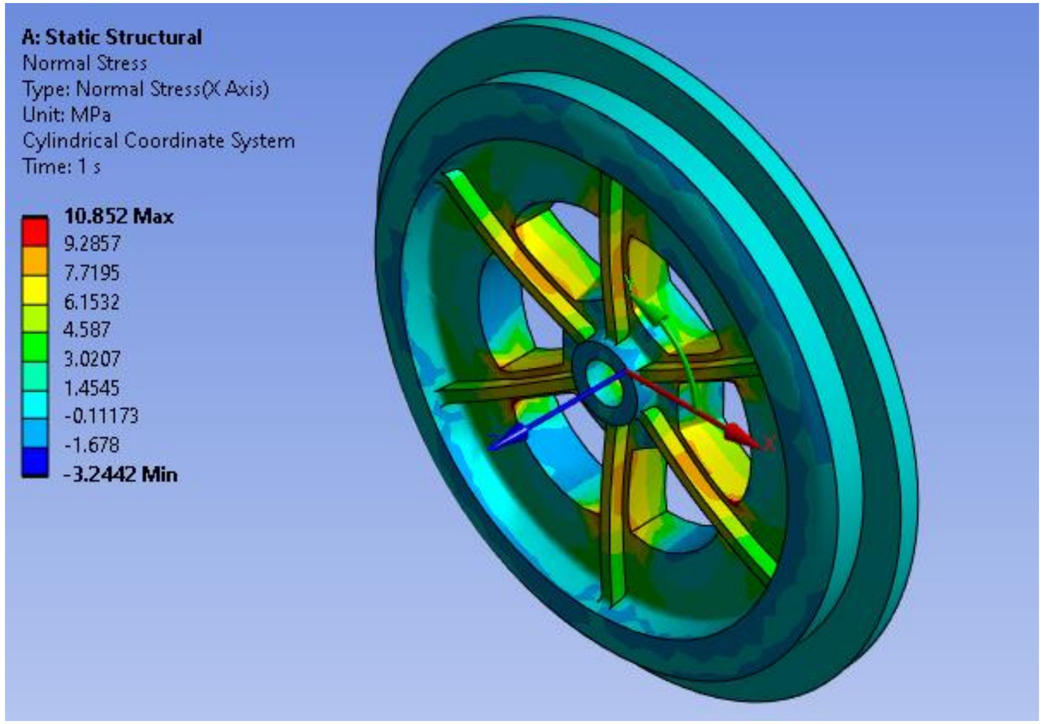


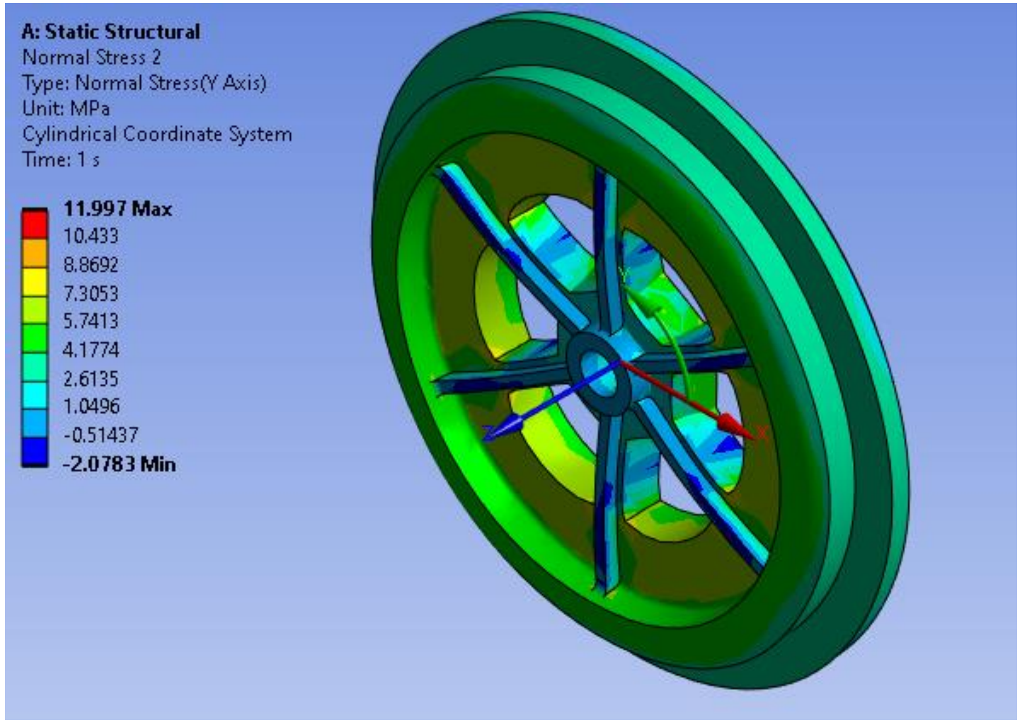


Workshop 01.1 – Static Postprocessing

- Insert Normal Stress results in the X and Y directions, again referencing the Cylindrical Coordinate System.
- Evaluate these results.

Click Here to see a demonstration of postprocessing static results



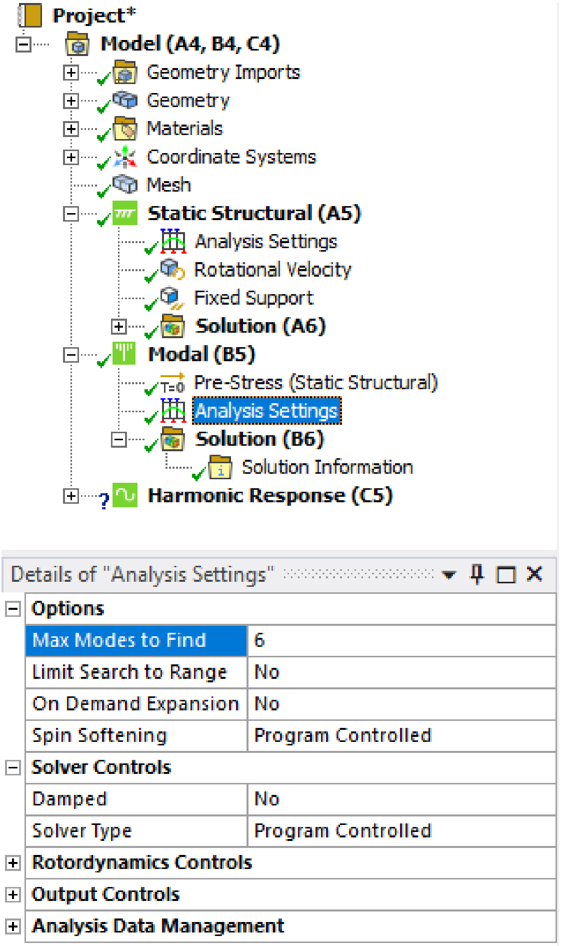


Workshop 01.1 – Modal Solution

- When performing pre-stressed modal analyses, the Modal branch of the model outline will inherit all the supports from the Static Structural branch.
  - Loads are not needed; the effect of the rotational velocity is accounted for by an increase in the stiffness of the structure resulting from its rotational velocity.
  - Thus, we don't have to apply any further loading information.
- Modal analyses are set up to extract the first six natural frequencies of the structure by default.
  - The subsequent harmonic analysis will use these frequencies and mode shapes to predict the response.
  - At this stage, we know that our harmonic acceleration load occurs over a frequency range of 0-500 Hz, but we don't know whether 6 natural frequencies will be enough to encompass that same range of excitation frequencies.
  - The accuracy of the harmonic analysis will rely on sufficient number of modes spanning the excitation frequency range.

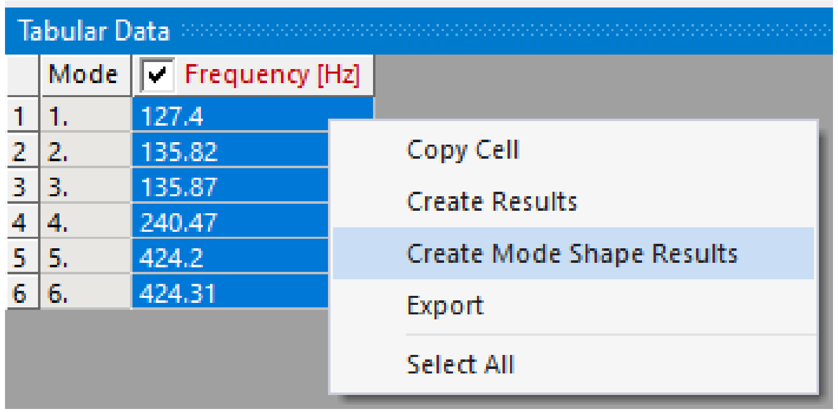
Workshop 01.1 – Modal Solution

- The Modal branch should appear as follows.
- Solve the Modal analysis.



Workshop 01.1 – Modal Postprocessing

- Results from a modal analysis consist of natural frequencies and mode shapes.
  - Each mode shape represents the deformation the structure would exhibit if it were vibrating at the corresponding frequency.
- Since our harmonic excitation will be along the Z axis, we would expect the maximum harmonic response to occur at a frequency(ies) that shows significant deformation of the structure in the Z direction.
  - These are known as resonant responses.
- Examine the deformation of each mode by creating Mode Shape Results.
  - Select the Solution branch under the Modal environment
  - Select the Frequency column of the Tabular Data
  - Right-mouse-button, Create Mode Shape Results
- Evaluate these results and review each one.



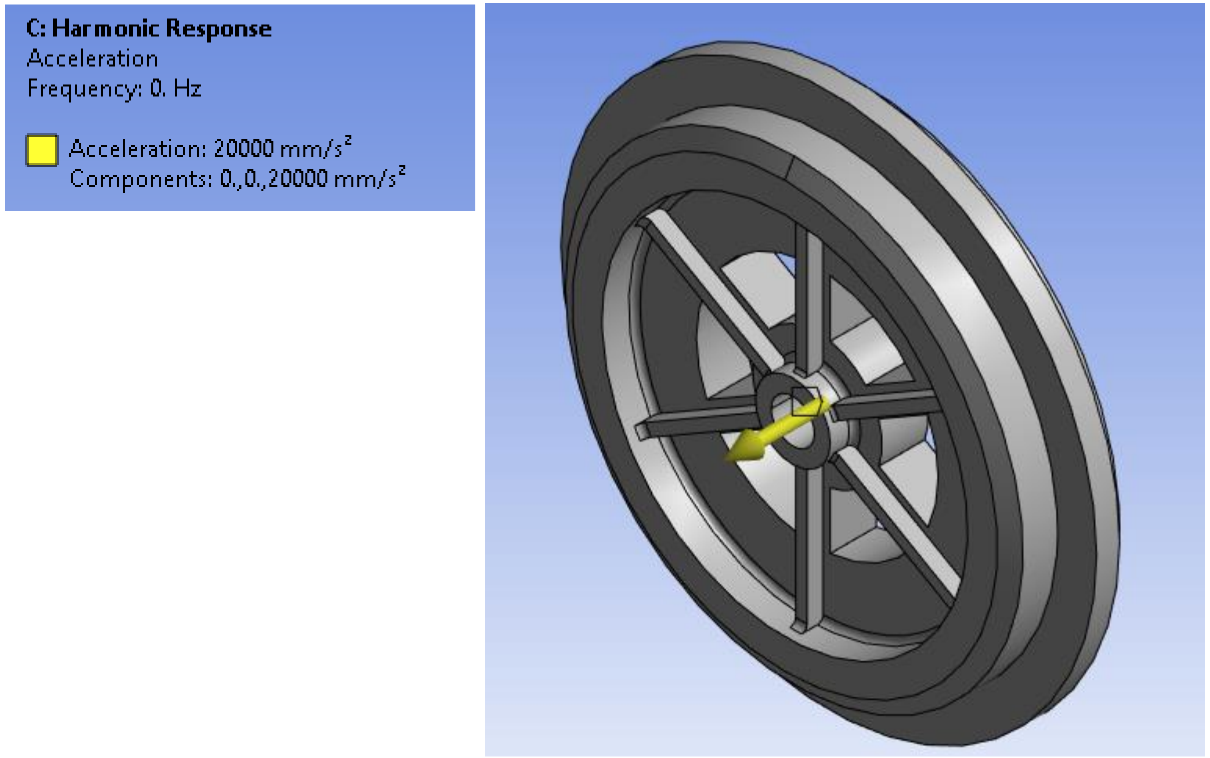
Workshop 01.1 – Modal Postprocessing

- Do you think there are other modes with frequencies higher than that of mode 6 within our 500 Hz excitation range?
- What mode(s) exhibit deformation in the Z direction?
  - Animating each mode shape may help in understanding the deformation shape.
  - Can you predict at what frequency the maximum harmonic response will occur?

Click Here to see a demonstration of the Modal analysis

Workshop 01.1 – Harmonic Preprocessing

- As with the modal analysis, the mode-superposition harmonic analysis will inherit all the supports from the modal analysis, and hence the static structural analysis.
- It remains to define the following for the harmonic analysis:
  - Acceleration load along the Z axis of 2G's (use 20,000 mm/s^2)



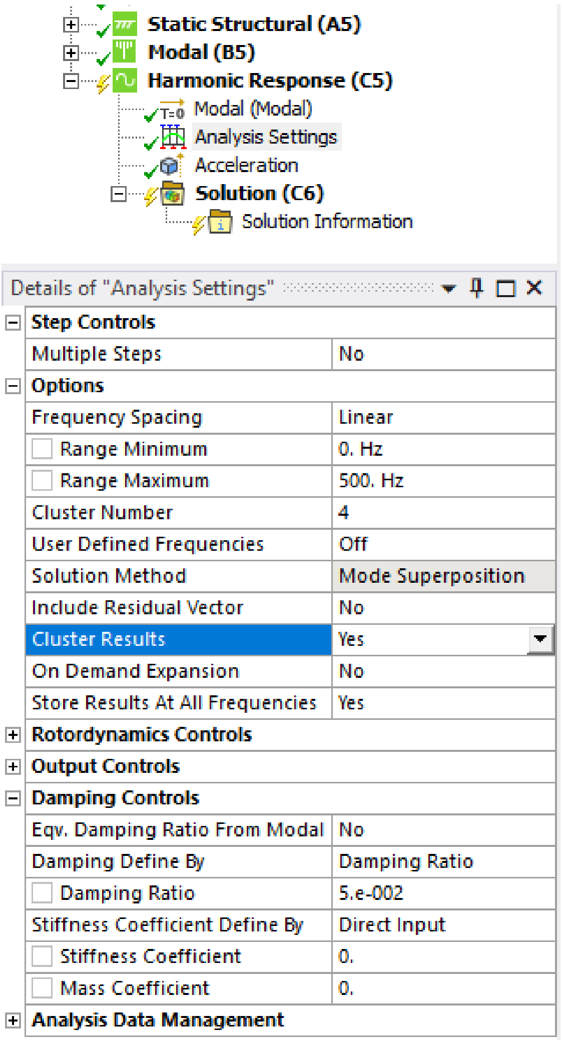
Workshop 01.1 – Harmonic Solution (Analysis Settings)

- All loads defined in a harmonic analysis are assumed to occur over a specified frequency range.
  - Define the minimum and maximum frequencies as 0 Hz and 500 Hz respectively.
- You define at what frequencies solutions are performed, either by:
  - Solution Intervals
    - Useful if you know the precise frequency at which you are interested in results
  - Cluster Results (results are clustered around each natural frequency)
    - Useful if you are looking to find peak responses at resonant conditions
    - **We'll use the cluster option to find peak responses**
- Damping is recommended, otherwise peak responses at resonant frequencies will be unbounded!
  - Apply damping as 5% Damping Ratio

Workshop 01.1 – Harmonic Solution (Analysis Settings)

- Your Analysis Settings inputs should look as follows:
- Note specifically,
  - Range Minimum = 0 Hz
  - Range Maximum = 500 Hz
  - Cluster Results = Yes
  - Damping Ratio = .05
- Solve the Harmonic Analysis

Click Here to see a demonstration of setting up the Harmonic analysis

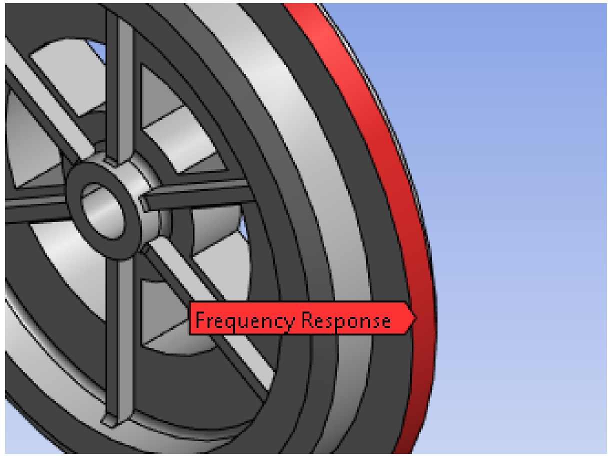


Workshop 01.1 – Harmonic Postprocessing

- Harmonic results usually consist of:
  - Frequency Response plots (amplitude vs. frequency) at discrete locations within the model
  - Contour results (deformation, stress) of the entire structure at the maximum response frequencies
- Knowing that our excitation was in the Z direction, we can expect most of our response to also be in the Z direction, as we had at least one mode shape that had a predominant Z component.
  - Insert a Frequency Response Deformation
    - Scope it to the outer rim of the flywheel
    - Set Spatial Resolution = Use Maximum
    - Orientation = Z axis
- Evaluate Results

Details of "Frequency Response"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Spatial Resolution	Use Average
Definition	
Type	Directional Deformation
Orientation	Z Axis
Coordinate System	Global Coordinate System
Suppressed	No
Options	
Results	

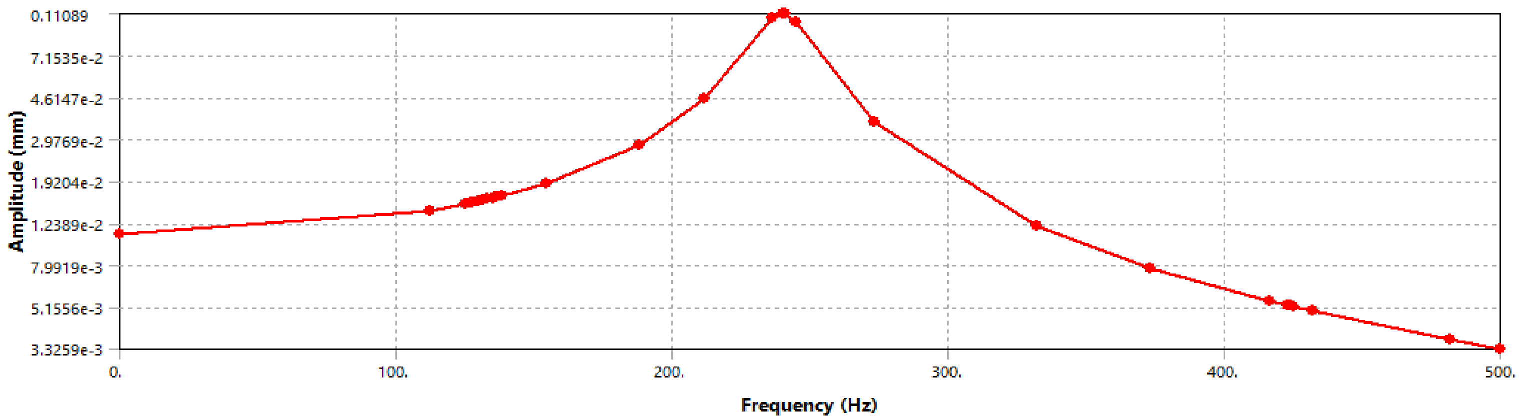


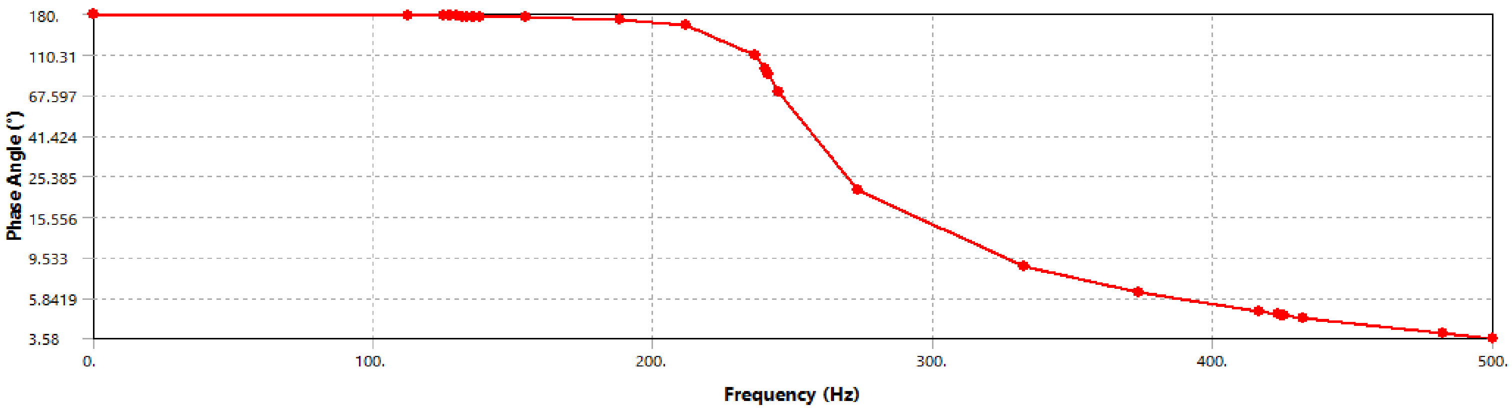
Workshop 01.1 – Harmonic Postprocessing

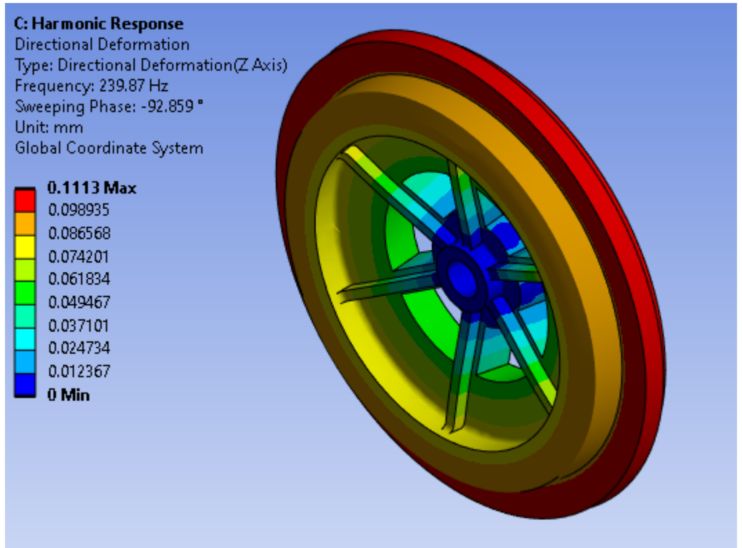
- Knowing the maximum displacement amplitude and the frequency at which it's occurring, use that information to generate contours of deformation and stress throughout the structure.
  - Right mouse button on the Frequency Response result and choose "Create Contour Result"
  - Evaluate the result
  - Note that the newly created Directional Deformation result is based upon the same Frequency and Sweeping Phase angle at which the maximum Frequency Response occurred.
  - Insert an Equivalent Stress result, once again evaluate it at the same Frequency and Sweeping Phase angle.
    - Copy/Paste the Frequency and Phase angle information from the Directional Deformation result.
- Your results should be similar to those on the next slide.

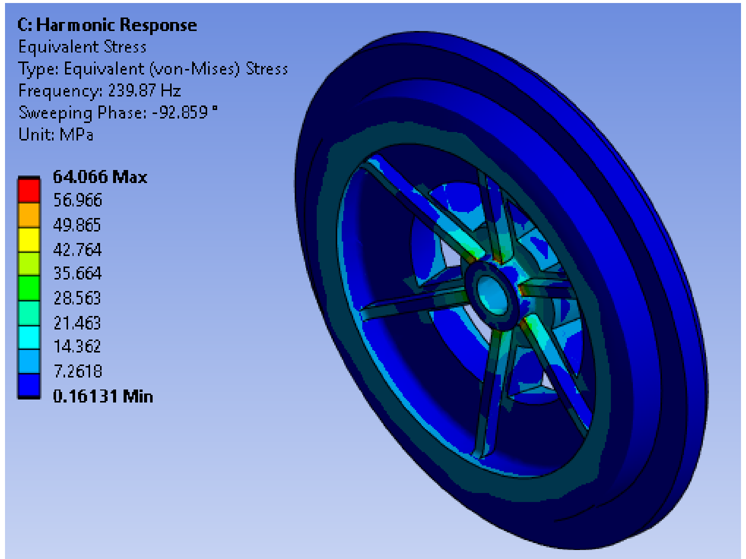
Workshop 01.1 – Harmonic Postprocessing

Frequency Response









Click Here to see a demonstration of postprocessing the harmonic analysis

Workshop 01.1 – Go Further

- Recall from slide 10 that the accuracy of the harmonic response calculations will depend upon a sufficient number of modes being extracted during the modal analysis.
  - Return to the Modal analysis branch and increase Max Modes to Find to 20
  - Solve the modal analysis once again
  - Solve the harmonic analysis once again
- Note any changes in the result
  - What was the maximum frequency obtained from the modal analysis?
  - Did the deformation and stress results from the harmonic analysis change significantly?

End of presentation