

ANSYS - Modal Analysis of a Composite Monocoque

Author: Jingsi Wu, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
 2. Geometry
 3. Mesh
 4. Physics Setup
 5. Numerical Solution
 6. Numerical Results
 7. Verification & Validation
- Exercises
Comments

This page has been moved to <https://courses.ansys.com/index.php/courses/modal-analysis-of-a-monocoque/>
Click in the link above if you are not automatically redirected in 10 seconds.



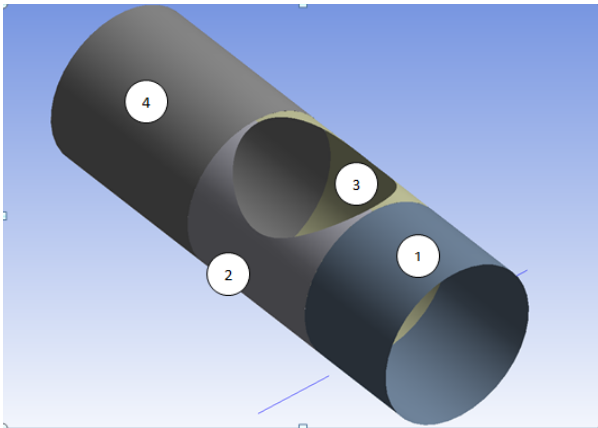
In order to do this exercise, you must have ANSYS Composite PrepPost installed.
All video tutorials contain audio instructions.

Modal Analysis of a Composite Monocoque

Created using ANSYS 14.0

Problem Specification

One of the important measure in race car design is torsional stiffness. To obtain that specification, a twist test must be performed on the chassis. The figure below shows a simplified version of a race car chassis with a composite monocoque and two front suspension links. To simulate a twist test, the back of the monocoque will be fixed while coupling will be applied to the end of the two front suspensions.



One of the beauty of composite material is to customize the ply lay-ups to achieve optimum properties according to the needs. It is often the case where different ply lay-ups are used at different monocoque regions for their specific needs. This exercise has 4 different regions as shown in the above diagram. Details of the composite layup will be explained during the exercise.

In this module, we will calculate torsional displacement with the use of ANSYS PrepPost and ANSYS Mechanical. Torsional stiffness will then be calculated using the following equations:

$$\theta = \frac{180}{\pi} \tan^{-1} \frac{\delta_t}{L}$$

$$\tau = F * L$$

$$k_t = \frac{\tau}{\theta}$$

θ : Angle of rotation

δ_t : Tangential displacement from Ansys

L : Length of the moment arm

F : Force applied

τ : Torque

K_t : Torsional Stiffness

[Go to Step 1: Pre-Analysis & Start-Up](#)

[Go to all ANSYS Learning Modules](#)