

ANSYS - Semi-monocoque shell - Problem Specification

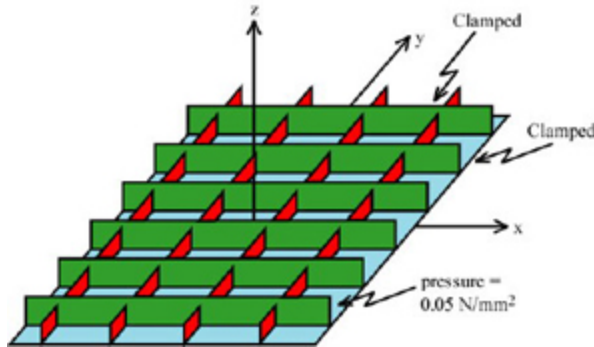
Author: Rajesh Bhaskaran, Cornell University

Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

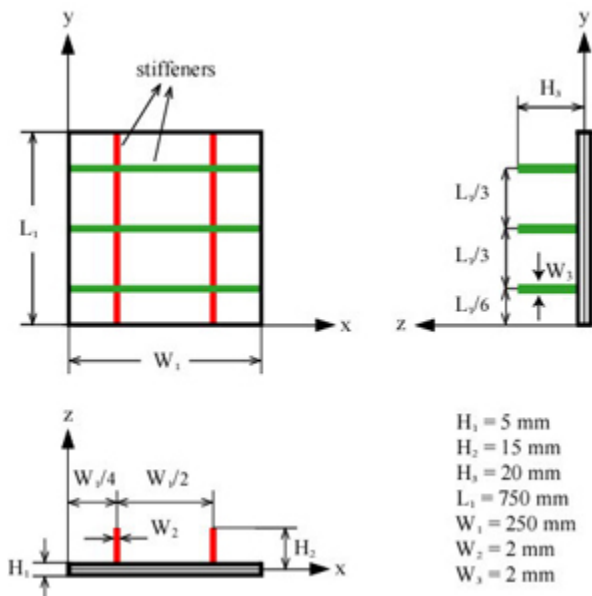
Problem Specification

In this tutorial, we'll analyze the plane shell with stiffeners shown in the figure below.



(Click for enlarged figure.)

All edges are clamped. A constant pressure of 0.05 N/mm^2 acts on the underside of the shell (i.e. the pressure acts in the +z direction). Since the geometry and loads are symmetric about both the x and y-axes, we need to model only one-quarter of the structure. The dimensions of the plate and stiffeners are shown in the figure below which shows only one-quarter of the structure.



The Young's modulus $E = 7.3 \times 10^4 \text{ MPa}$ and the Poisson ratio is 0.33.

Print out this page and have it on hand so that you can easily refer to the geometry and problem specification as you go through the tutorial.

Go to [Step 1: Start-up and preliminary set-up](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)