

ANSYS 12 - LaminatePlate - Pre-Analysis & Start-Up



This Tutorial is Under Construction!

Author: John Singleton, Cornell University

Problem Specification

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

1. Pre-Analysis & StartUp

Open ANSYS Workbench

Find where the ANSYS shortcut is located and launch ANSYS Workbench

Static Structural Analysis System

The problem at hand is a static structural problem so drag **Static Structural (ANSYS)** over to the project schematic. Rename the stand alone system to "VM82LaminatePlate".

Engineering Data

By default for Static Structural problems ANSYS uses the material properties of structural steel. For this laminate plate problem the pre-assigned material properties will be over written by interjecting command "snippets" into ANSYS workbench. Thus, no changes need to be made within **Engineering Data**.

Save

Save your project as "LaminatePlate" and don't forget that in order to open the project in the future you will need both the "LaminatePlate.wbpj" file and the "LaminatePlate_files" folder.

[Go to Step 2: Geometry](#)

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

[Go to all ANSYS Learning Modules](#)