

ANSYS 12 - LaminatePlate - Mesh



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

3. Mesh

Launch Mechanical

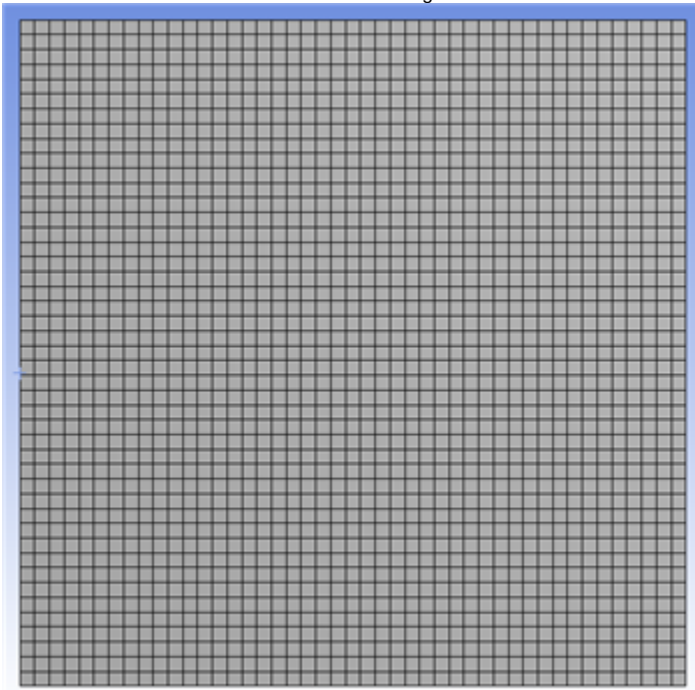
In order to start Mechanical *(Double Click) model*. After Mechanical opens, click on the Z vector in the triad to properly orient the plate.

Mapped Face Meshing

For the laminated plate problem a grid mesh will be used such that the areas of the elements are all equivalent. In order to create a mapped face mesh *(Right Click) Mesh > Insert > Mapped Face Meshing*. Next, apply the surface of the plate as the geometry for the mapped face meshing.

Edge Sizing

In order to control the element size and consequently the number of elements in the mesh, edge sizing will be used. To create an edge sizing control *(Right Click) Mesh < Insert < Sizing*. Next, apply all four edges of the plate as the geometry for the sizing. Then, *(Right Click) Mesh > Generate Mesh* to create the mesh. You should obtain the following mesh.



Leave mechanical open for the next step.

[Go to Step 4: Setup \(Physics\)](#)

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

[Go to all ANSYS Learning Modules](#)