# **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**

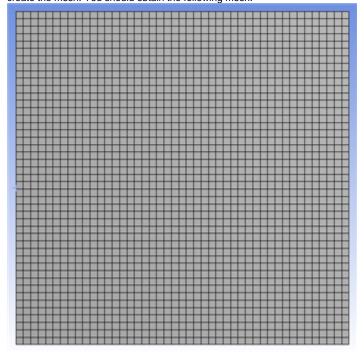
In order to start Mechanical (Double Click) model. After Mechanical opens, click on the Z vector in the triad to properly orient the plate.

## **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 (*Ri ght 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