

ANSYS 12 - LaminatePlate - Geometry



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](#)

2. Geometry

Launch Design Modeler

In order to start the Design Modeler (**Double Click**) *Geometry*. After the Design Modeler opens, select meter as the desired length unit.

Proper Orientation

The sketching will be done in the XY plane, so (**Click**) *XY Plane*, then click on the face plane button, .

Line Sketching

The laminate plate will be modeled using shell elements, thus the geometry needed will simply be a square in the XY Plane. In order to draw the rectangle (**Click**) *Sketching* > (**Click**) *Rectangle* then draw a rectangle in the XY Plane.

Dimensioning

Dimension the rectangle such that both sides are defined to be 10m.

Surface Creation

Here, the rectangle will be turned into a surface. To do so (**Click**) *Concept* > *Surface From Sketches*. Next, highlight all four edges of the rectangle and select **Apply** in the "Details of SurfaceSK1" table. Now, click the generate button,  **Generate**, in order to create the surface.

[Go to Step 3: Mesh](#)

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

[Go to all ANSYS Learning Modules](#)