

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](#)