Module 9: Stresses in a Symmetric Cross-Ply Laminar Composite in Tension

Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Problem Description</td>
<td>2</td>
</tr>
<tr>
<td>Theory</td>
<td>3</td>
</tr>
<tr>
<td>Geometry</td>
<td>6</td>
</tr>
<tr>
<td>Preprocessor</td>
<td>8</td>
</tr>
<tr>
<td>Element Type</td>
<td>8</td>
</tr>
<tr>
<td>Material Properties</td>
<td>9</td>
</tr>
<tr>
<td>Section Properties</td>
<td>10</td>
</tr>
<tr>
<td>Meshing</td>
<td>12</td>
</tr>
<tr>
<td>Loads</td>
<td>13</td>
</tr>
<tr>
<td>Solution</td>
<td>14</td>
</tr>
<tr>
<td>General Postprocessor</td>
<td>15</td>
</tr>
<tr>
<td>Results</td>
<td>18</td>
</tr>
<tr>
<td>Validation</td>
<td>19</td>
</tr>
</tbody>
</table>
**Problem Description**

**Nomenclature:**
- $[0/90]_s$: Stacking Sequence
- $E_1 = 20 \times 10^6$ psi: Young’s Modulus in the Fiber Direction
- $E_2 = 1.3 \times 10^6$ psi: Young’s Modulus Perpendicular to the Fiber Direction
- $G_{12} = 1.03 \times 10^6$ psi: X-Y Plane Shear Modulus
- $h = 12t$ in: Total Thickness
- $L \times W = 1 \times 1$ in: Composite Length and Width
- $N_x = 1000$ lbf: Pull Force
- $t = 0.005$ in: Top and Bottom Layer Thickness
- $T_{cure} = 370$ °F: Curing Temperature
- $T_{ref} = 70$ °F: Room Temperature
- $\alpha_1 = -0.17 \mu\text{in/}^{\circ}\text{F}$: Thermal Expansion Coeff. in the Fiber Direction
- $\alpha_2 = 15.57 \mu\text{in/}^{\circ}\text{F}$: Thermal Expansion Coeff. Perpendicular to the Fiber Direction
- $\nu_{12} = 0.3$: X-Y Plane Poisson’s Ratio

In this module, we will be modeling the stresses in a Symmetric Cross-Ply Laminar Composite in tension subject to residual thermal stresses in ANSYS Mechanical APDL. This composite consists of four layers, thin outer layers with fibers oriented along the x-axis and thicker inner layers with fibers oriented along the y-axis. The example was pulled from “Laminar Composites” a graduate level textbook by George H. Stabb. Our model will use two dimensional layered shell elements. We will compare the results with the analytical solution based on the Generalized Hooke’s Law. This module will emphasize techniques on modeling orthotropic materials and layered materials without creating complicated CAD files.
The Generalized Hooke’s Law relates stress and strain of any material in the following way:

\[ \sigma_i = C_{ij}\varepsilon_j \quad 1 \leq i, j \leq 6 \]  

(9.1a)

Where \( C_{ij} \) is the stiffness matrix. This equation can also be written:

\[ \varepsilon_i = S_{ij}\sigma_j \]  

(9.1b)

Where \( S_{ij} \) is the compliance matrix.

An orthotropic material has material properties that vary along perpendicular directions at a point in the body resulting in 3 planes of material symmetry. Thus, orthotropic material properties vary as a function of orientation. The Generalized Hooke’s Law for an Orthotropic Material reduces to:

\[
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\varepsilon_3 \\
\gamma_{23} \\
\gamma_{31} \\
\gamma_{12}
\end{bmatrix} =
\begin{bmatrix}
\frac{1}{E_1} & -\frac{v_{12}}{E_2} & -\frac{v_{13}}{E_3} & 0 & 0 & 0 \\
-\frac{v_{12}}{E_1} & \frac{1}{E_2} & -\frac{v_{23}}{E_3} & 0 & 0 & 0 \\
-\frac{v_{13}}{E_1} & -\frac{v_{23}}{E_2} & \frac{1}{E_3} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{G_{23}} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{G_{31}} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1}{G_{12}}
\end{bmatrix}
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\sigma_3 \\
\tau_{23} \\
\tau_{31} \\
\tau_{12}
\end{bmatrix}
\]  

(9.1c)

The material symmetry of orthotropic materials can be explained as

\[ S_{ij} = S_{ji} \]  

(9.2a)

or

\[ \frac{v_{ij}}{E_i} = \frac{v_{ji}}{E_j} \]  

(9.2b)

Additionally, if we assume that the fibers bear load in the z direction and are isotropic, then all properties in the z direction are equal to the properties in the x direction. We can thus derive the following relationships:

\[ E_1 = E_3 \]
\[ v_{23} = \frac{E_2}{E_3} v_{32} \]
\[ v_{12} = v_{32} \]
\[ G_{12} = G_{23} = G_{31} \]  

(9.3)
These relationships are important when we input material properties into ANSYS since the stiffness matrix must be positive definite, however for the theory we will assume plane stress and plane strain conditions (all ‘3’ terms are neglected).

The in the Generalized Hooke’s Law for Plane Stress, \( C_{ij} \) is replaced by \( Q_{ij} \) \( 1 \leq i, j \leq 3 \).

\[
Q_{11} = \frac{S_{22}}{S_{11}S_{22} - S_{12}^2} = \frac{E_1}{1 - \nu_{12}\nu_{21}} \\
Q_{12} = -\frac{S_{12}}{S_{11}S_{22} - S_{12}^2} = \frac{\nu_{12}E_2}{(1 - \nu_{12}\nu_{21})} = \frac{v_{21}E_1}{1 - \nu_{12}\nu_{21}} \\
Q_{22} = \frac{S_{11}}{(S_{11}S_{22} - S_{12}^2)} = \frac{E_2}{1 - \nu_{12}\nu_{21}} \\
Q_{33} = \frac{1}{S_{66}} = G_{12}
\] (9.4)

For our problem:

\[
\begin{bmatrix}
20.14 & 0.392 & 0 \\
0.392 & 1.307 & 0 \\
0 & 0 & 1.03
\end{bmatrix} * 10^6 \text{psi} \quad \begin{bmatrix}
1.307 & 0.392 & 0 \\
0.392 & 20.14 & 0 \\
0 & 0 & 1.03
\end{bmatrix} * 10^6 \text{psi}
\]

Since we have both thermal and tensile loads, let

\( \tilde{N} = N_T + N_x \)

\( \tilde{M} = M_T + 0 \) (9.5)

Where \( \tilde{M} \) are the bending moments caused by the coupling of extension and bending.

Combining thermal and tensile effects, the equation for mid surface strains becomes:

\[
\begin{bmatrix}
\epsilon^0 \\
\kappa
\end{bmatrix} = \begin{bmatrix}
A' & B' \\
B' & D'
\end{bmatrix} \begin{bmatrix}
\tilde{N} \\
\tilde{M}
\end{bmatrix}
\] (9.6a)

Where \( A' \) is the compliance due to \( Q \), \( B' \) includes coupling effects, and \( D' \) terms include flexural rigidity for bending. Since the laminate is symmetric, no bending occurs. Thus, \( \tilde{M} = B' = \kappa = 0 \)

\[
\{\epsilon^0\} = \{A'\}\{\tilde{N}\}
\] (9.6b)

Since we can neglect \( \tilde{M} \), \( D' \) does not need to be calculated. To calculate \( A' \), we must invert \( A_{ij} \):

\[
A_{ij} = \Sigma [Q]_{ik} t_k
\] (9.7a)

Or

\[
A_{ij} = (2[Q]_0 + 10[Q]_{90}) t = \begin{bmatrix}
4.45 & 0.392 & 0 \\
0.392 & 17 & 0 \\
0 & 0 & 1.03
\end{bmatrix} * 10^6 h
\] (9.7b)
Inverting $A_{ij}$ we get:

$$
A' = \begin{bmatrix}
.226 & -.005 & 0 \\
-.005 & .0589 & 0 \\
0 & 0 & .972
\end{bmatrix} \times \left( \frac{10^{-6}}{h} \right) \tag{9.8}
$$

This composite was cured at 370°F but is sitting in a room at 70°F. Thus we must consider the residual stresses resulting from the temperature difference.

$$
\Delta T = T_{ref} - T_{cure} = -300^\circ F \tag{9.9}
$$

Using the Thermal Expansion Coefficients specified in the problem:

$$
\{\alpha_0\} = \begin{bmatrix} -1.17 \\ 15.57 \\ 0 \end{bmatrix} \times 10^{-6} \text{ in/in}^\circ F \quad \{\alpha_{90}\} = \begin{bmatrix} 15.57 \\ -1.17 \\ 0 \end{bmatrix} \times 10^{-6} \text{ in/in}^\circ F \tag{9.10}
$$

The thermal loads are expressed as:

$$
\{N_T\} = \sum [Q]_{k} \{\alpha\}_k t_k \Delta T = (2[Q]_0 + 10[Q]_{90}) t \Delta T 
$$

$$
\{N_T\} = \begin{bmatrix} 17.4 \\ 5.69 \\ 0 \end{bmatrix} h \Delta T = \begin{bmatrix} 313 \\ 102 \\ 0 \end{bmatrix} \text{ lbf} \tag{9.11}
$$

Substituting into eqn 9.6b, we get

$$
\begin{bmatrix}
\varepsilon_x^0 \\
\varepsilon_y^0 \\
\gamma_{xy}^0
\end{bmatrix} = \begin{bmatrix}
.226 & -.005 & 0 \\
-.005 & .0589 & 0 \\
0 & 0 & .972
\end{bmatrix} \begin{bmatrix}
N_x - 313 \\
-102 \\
0
\end{bmatrix} \times \left( \frac{10^{-6}}{h} \right) = \begin{bmatrix}
3.76N_x - 1170 \\
-.0867N_x - 73.5 \\
0
\end{bmatrix} \times 10^{-6} \tag{9.12}
$$

The stresses in each lamina, $k$, is:

$$
\{\sigma\}_k = [Q]_k ([e^0] - \{\alpha\}_k \Delta T) \tag{9.13}
$$

Thus

$$
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}_o = \begin{bmatrix} 75.7N_x - 22800 \\ 1.36N_x + 5550 \\ 0 \end{bmatrix} \quad \begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}_{\alpha_{90}} = \begin{bmatrix} 4.88N_x + 4550 \\ -.271N_x - 1120 \\ 0 \end{bmatrix} \tag{9.14}
$$

Substituting $N_x$:

$$
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}_o = \begin{bmatrix} 52900 \\ 6800 \\ 0 \end{bmatrix} \quad \begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}_{\alpha_{90}} = \begin{bmatrix} 9430 \\ -1391 \\ 0 \end{bmatrix} \tag{9.15}
$$
**Geometry Preferences**

1. Go to **Main Menu -> Preferences**
2. Check the box that says **Structural**
3. Click **OK**
Square

The geometry for this problem is irrelevant as long as it is either square or rectangular since it does not affect the physics involved. Thus, we will model a unit square.

1. Go to Main Menu -> Preprocessor -> Modeling -> Create -> Areas -> Rectangle -> by Dimensions
2. Under X1,X2 X-coordinates enter 0 and 1 in the two fields
3. Under Y1,Y2 Y-coordinates enter 0 and 1 into the two fields
4. Press OK

The geometry produced is shown below:

Saving Geometry

It would be convenient to save this model so that it does not have to be made again from scratch.

1. Go to File -> Save As …
2. Under Save Database to pick a name for the Geometry. For this tutorial, we will name the file Cross-Ply Composite
3. Under Directories: pick the Folder you would like to save the .db file to.
4. Click OK
Preprocessor

Element Type

1. Go to Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete
2. Click Add
3. Click Shell -> 4node181
4. Click OK

SHELL181 has the capability of modeling laminar composites up to 255 layers. We will modify the properties of each layer in the Section Properties section. The advantage of using layered shell elements is great since a complex CAD model with fibers need not be constructed. For more information on SHELL181, visit the ANSYS HELP.

By default, SHELL181 only stores results of the Top, Middle, and Bottom Lamina. Since we have 4 layers, we must change the settings to store information on all Lamina.

5. Go to Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete
6. Click Options
7. Under Storage of Layer Data
   Pick All layers
8. Click OK
9. Click Close
Material Properties

1. Go to Main Menu -> Preprocessor -> Material Properties -> Material Models ->
   Structural -> Linear -> Elastic -> Orthotropic
2. Fill in the values as listed in the theory section
3. Click OK

4. Go to Main Menu -> Preprocessor -> Material Properties -> Material Models ->
   Structural -> Thermal Expansion -> Secant Coefficient -> Orthotropic ->
5. Under reference temperature, enter the Temperature at which there are no thermal strains. For us, that is the curing temperature, 370
6. Enter the directional Secant Coefficients as shown.
7. Click OK
8. Exit out of Define Material Model Behavior window
Section Properties

The layers of SHELL181 are defined starting from the bottom layer up. We must define the lamina in sections.

1. Go to **Main Menu -> Preprocessor -> Sections -> Shell -> Lay-up -> Add / Edit**
2. Under **Thickness** enter **0.005**
3. Under **Material ID** enter 1. That is the material we have defined on pg 9.
4. Under **Orientation** enter 0 for 0 degrees from the X-axis
5. Under **Integration Pts** put 3. This locates an integration point at the top, middle, and bottom of each layer. Since we are solving a linear problem, 3 integration points will give us more than enough accuracy to solve the problem.
6. Click **Add Layer**
7. Repeat steps 3-5 with the following values: **0.025,1,90,3**
8. Since the Laminate is Symmetric, we can mirror the Layup for the top layers. Go to **Create and Modify Shell Sections -> Tools -> Add Symmetry**

The Create and Modify Shell Sections menu should look as follows:

9. Click **OK**
10. Go to **Main Menu -> Preprocessor -> Sections -> Shell -> Lay-up -> Plot Section**
11. Under **Plot Section with ID** select 1
12. Under **LAYR1, LAYR2** enter 1, 4 into the two fields
13. Click **OK**

The section plot populates as shown below:

![Section Plot](image)

We have thus created the cross section of our plate with all of the relevant material properties data called to one window. There is no need to model complicated CAD files with fibers, since the section properties can be accounted for easily in the Preprocessor.
Meshing

1. Go to **Main Menu -> Preprocessor -> Meshing -> Mesh Tool**
2. Go to **Size Controls: -> Global -> Set**
3. Under **SIZE Element edge length** put 0.5. This will create a mesh of 4 square elements.
4. Click **OK**
5. Click **Mesh**
6. Click **Pick All**
7. Click **Close**
8. Go to **Utility Menu -> SAVE_DB**

The resulting mesh looks as shown:
Loads

Thermal Lead

1. Go to Main Menu -> Preprocessor -> Loads -> Settings -> Uniform Temp
2. Under Uniform Temperature enter 70
3. Click OK

DOF Constraint

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Displacement -> On Nodes
2. Click the Node at the center of the plate and press OK
3. Under DOFs to be Constrained select All DOF
4. Under Value enter 0
5. Click OK
Tensile Load

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads ->
   Apply -> Structural -> Pressure -> On Lines
2. Select the left and right lines of the domain
3. Click OK
4. Under VALUE put -1000
5. Click OK

Solution

   This step may take some time depending on mesh size and the speed of your computer
   (generally a minute or less).
General Postprocessor

Deformed Shape

Before collecting data, it is wise to check the deformed shape of the plate to check if the qualitative answer makes sense with the physics of the problem.

1. Go to Main Menu -> General Postproc -> Plot Results -> Deformed Shape
2. Click Def + undeformed
3. Click OK

This will plot the deformed shape with respect to the original undeformed shape.

As we can see, the plate extends in the X direction and contracts slightly in the Y direction. From a qualitative perspective, this makes sense since we are pulling on the plate in the X direction, causing Poisson’s Effect contraction in the Y direction. In addition, due to thermal effects, it makes sense the plate would shrink in the Y direction since the positive thermal expansion coefficient of the matrix is several orders of magnitude larger than the negative thermal expansion coefficient of the carbon fibers.
Layer 1 Stresses

ANSYS defaults to viewing the results of the top and bottom layers, so we will record the stress values at these layers first.

1. Go to **Main Menu -> General Postproc -> Plot Results -> Contour Plot -> Nodal Solution**
2. Go to **Nodal Solution -> Stress -> X-Component of stress**
3. Click **OK**

The following plot should populate:

![Contour Plot](image)

The result for stress in the X direction for the top and bottom lamina is **52916 psi** as boxed in red. This procedure can be repeated for stress in the Y and X directions, to which ANSYS yields the answer **6897 psi** and **0 psi** respectively.

**USEFUL TIP:** If you want to recreate these color inverted plots, go to **Utility Menu -> PlotCtrls -> Write Metafile -> Invert White / Black.** This creates a png capture of the current ANSYS window image with inverted white and black colors.
Layer 2 Stresses

On page 8, we instructed the program to record data for all layers. To view the data on different layers, we must instruct the program to do so.

1. Go to Utility Menu -> ANSYS Command Prompt
2. Type the command `Layer,2` and press `enter`. This will let us view the results of layer 2.

3. Go to Main Menu -> General Postproc -> Plot Results -> Contour Plot -> Nodal Solution
4. Go to Nodal Solution -> Stress -> X-Component of stress
5. Click OK

The following plot should populate:

The result for stress in the X direction for the middle layers is 9417 psi as boxed in red. This procedure can be repeated for stress in the Y and X directions, to which ANSYS yields the answer -1379 psi and 0 psi respectively.
**Results**

The percent error (%E) in our model can be defined as:

\[
%E = \text{abs} \left( \frac{\sigma_{\text{theoretical}} - \sigma_{\text{model}}}{\sigma_{\text{theoretical}}} \right) \times 100
\]  

(9.15)

<table>
<thead>
<tr>
<th>Stress</th>
<th>4 Elements (%E)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\sigma_{x1})</td>
<td>0.03</td>
</tr>
<tr>
<td>(\sigma_{y1})</td>
<td>1.43</td>
</tr>
<tr>
<td>(\sigma_{z1})</td>
<td>0.00</td>
</tr>
<tr>
<td>(\sigma_{x2})</td>
<td>0.14</td>
</tr>
<tr>
<td>(\sigma_{y2})</td>
<td>0.86</td>
</tr>
<tr>
<td>(\sigma_{z2})</td>
<td>0.00</td>
</tr>
</tbody>
</table>

According to the results, the answers for 4 elements are accurate within 1.43% of the theoretical solution. Thus, for a plane stress problem, the answer can be reached within a very coarse mesh. In the following section we will learn that further mesh refinement does not add accuracy to the answer. The problem has fully converged after 4 elements! In the theory section, our answer was calculated with some rounding error, so it is possible that the answer ANSYS has calculated may be more accurate than the results in the theory section!
After our mesh refinement study, we learned that the mesh was perfectly converged after 4 elements. Since each layer had 3 integration points, each layer was 6th order accurate. Since axial stresses and thermal stresses are linear, there are no approximations in the numerical technique. Thus the answer provided would be exact given any mesh size. Try to look out for these scenarios as they will save you mesh size, run time, and memory in the future.