1. xxxxx

The local coordinate naming rule in the composite world is not arbitrary like other static analysis. For a fiber-reinforced composite material which is the main subject of this manual, as Figure 1 shows, “1” is the longitudinal direction of the fiber material in a composite, “2” is the transverse direction of the fiber material, and “3” is orthogonal to “1-2” plane in a way similar to traditional Cartesian coordinates.

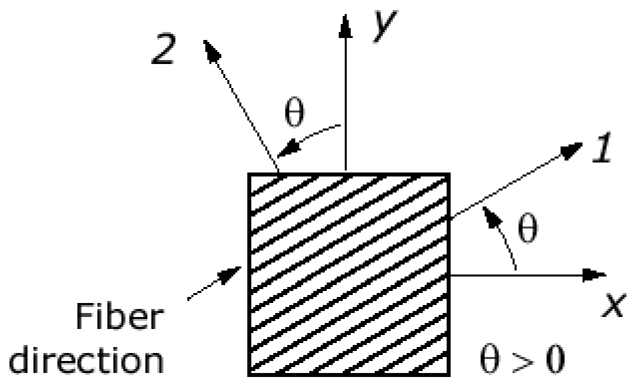

Figure 1. Showing the relation between 1-2 Local coordinate system and x-y global coordinate system.

This should be obvious that composites are not isotropic materials. Therefore, the governing properties for a plane stress analysis need to be changed accordingly. A laminate has 6 different Poisson’s ratios, $v_{12}, v_{13}, v_{23}, v_{21}, v_{31}, v_{32}$. The first three are major Poisson’s ratio and the other three are minor Poisson’s ratio. Major Poisson’s ratio is related to minor Poisson’s ratio according to

$$\frac{v_{ij}}{E_i} = \frac{v_{ji}}{E_j}, \tag{1.1}$$

where E_i is the elastic modulus of a composite. Usually all the properties are given, but if they are not given anywhere, only four properties, E_1, E_2, v_{12}, G_{12}, need to be measured.

The transformation from local to the global Cartesian system can be expresses as

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = [T]^{-1} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix}, \tag{1.2}$$

where [T] is the transformation matrix

$$[T] = \begin{bmatrix} cos^2(\theta) & sin^2(\theta) & 2\cos\theta\sin\theta \\ sin^2(\theta) & cos^2(\theta) & -2\cos\theta\sin\theta \\ -\cos\theta\sin\theta & \cos\theta\sin\theta & cos^2(\theta) - sin^2(\theta) \end{bmatrix}, \tag{1.3}$$

And θ is the fiber direction with respect to the x-axis.

2. *Lamina*

In a static analysis, the stress-strain relationship of a lamina is governed by

$$\{\sigma\} = [D]\{\varepsilon\}, \tag{2.1}$$

where $[D]$ is a 6x6 matrix in typical 3D analysis. However, fiber composites are considered to be orthotropic, and equation 2.1 can be simplified to

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{21} & Q_{22} & 0 \\ 0 & 0 & 2Q_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \frac{\gamma_{12}}{2} \end{Bmatrix}. \tag{2.2}$$

This 3x3 matrix is call the lamina stiffness matrix. The components inside this matrix can be calculated through these equations,

$$Q_{11} = \frac{E_1}{1-v_{12}v_{21}}, \tag{2.3}$$

$$Q_{12} = Q_{21} = \frac{v_{12}E_2}{1-v_{12}v_{21}}, \tag{2.4}$$

$$Q_{22} = \frac{E_2}{1-v_{12}v_{21}}, \tag{2.5}$$

$$Q_{66} = G_{12}. \tag{2.6}$$

However, the stiffness matrix calculated here is based on local coordinates. Thus, a local to global transformation is necessary. After the transformation can be written as

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & 2\bar{Q}_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \frac{\gamma_{xy}}{2} \end{Bmatrix}. \tag{2.7}$$

Substitute equation 1.3 into 2.7, the formulas for each component are

$$\bar{Q}_{11} = Q_{11}c^4 + Q_{22}s^4 + 2(Q_{12} + 2Q_{66})s^2c^2, \tag{2.8}$$

$$\bar{Q}_{12} = (Q_{11} + Q_{22} - 4Q_{66})s^2c^2 + Q_{12}(c^4 + s^4), \tag{2.9}$$

$$\bar{Q}_{22} = Q_{11}s^4 + Q_{22}c^4 + 2(Q_{12} + 2Q_{66})s^2c^2, \tag{2.10}$$

$$\bar{Q}_{16} = (Q_{11} - Q_{12} - 2Q_{66})sc^3 - (Q_{22} - Q_{12} - 2Q_{66})cs^3, \tag{2.11}$$

$$\bar{Q}_{26} = (Q_{11} - Q_{12} - 2Q_{66})cs^3 - (Q_{22} - Q_{12} - 2Q_{66})sc^3, \quad (2.12)$$

$$\bar{Q}_{66} = (Q_{11} + Q_{22} - 2Q_{12} - 2Q_{66})s^2c^2 + Q_{66}(c^4 + s^4), \quad (2.13)$$

and c, s are cosine and sine of the angle between "1" direction and X direction.

3. *Laminate*

For a laminate, the origin of the global Cartesian coordinate locals at the middle of it, instead of at the top or bottom. Since the orientation starts at the middle, the distance z outmost lamina to the origin can be positive or negative depending on the Z axis. The layer on the top is layer one and goes to layer N at the bottom. For more naming and orientation code, please refer to page 300 in Principles of Composite Material Mechanics by Ronald F. Gibson[1].

ADB stiffness matrix

Stress analysis for laminate is more complicated than that of lamina. Since there are more than one layer, three stress resultants should be taken into consideration. The laminate extensional stiffnesses are given by

$$A_{ij} = \int_{-t/2}^{t/2} \left(\bar{Q}_{ij}\right)_k dz = \sum_{k=1}^{N} \left(\bar{Q}_{ij}\right)_k (z_k - z_{k-1}). \quad (3.1)$$

The laminate-coupling stiffnesses are given by

$$B_{ij} = \int_{-t/2}^{t/2} \left(\bar{Q}_{ij}\right)_k zdz = \frac{1}{2} \sum_{k=1}^{N} \left(\bar{Q}_{ij}\right)_k (z_k^2 - z_{k-1}^2), \quad (3.2)$$

and the laminate-bending stiffnesses are given by

$$C_{ij} = \int_{-t/2}^{t/2} \left(\bar{Q}_{ij}\right)_k z^2 dz = \frac{1}{3} \sum_{k=1}^{N} \left(\bar{Q}_{ij}\right)_k (z_k^3 - z_{k-1}^3). \quad (3.3)$$

A complete set of equations can be expressed as

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} \\ A_{16} & A_{26} & A_{66} & B_{16} & B_{26} & B_{66} \\ B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{12} & B_{22} & B_{26} & D_{12} & D_{22} & D_{26} \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}. \quad (3.4)$$

[1] R. F. Gibson, *Principles of composite material mechanics*, 4th ed. Boca Raton: CRC Press, Taylor & Francis Group, 2016.

Do not get scared by this large matrix. For a laminate with a symmetry stacking order, [B] matrix is just zero. MATLAB code for this calculation is also included in the Appendix A.

4. *Failure criteria*

There are many ways to define failure for composite materials. The most direct way is comparing stress and strain resultant with material strength. The governing equations are

$$-s_L^{(-)} < \sigma_1 < s_L^{(+)}, \tag{4.1}$$

$$-s_T^{(-)} < \sigma_2 < s_T^{(+)}, \tag{4.2}$$

$$|\tau_{12}| < s_{LT}, \tag{4.3}$$

$$-e_L^{(-)} < \varepsilon_1 < e_L^{(+)}, \tag{4.4}$$

$$-e_T^{(-)} < \varepsilon_2 < e_T^{(+)}, \tag{4.5}$$

$$|\gamma_{12}| < e_{LT}, \tag{4.6}$$

where $s_L^{(-)}, s_T^{(-)}$ are compressive strengths in longitudinal and transvers direction, $s_L^{(+)}, s_T^{(+)}$ are tensile strengths in longitudinal and transvers direction, s_{LT} is the effective shear strength, and $e_L^{(-)}, e_T^{(-)}, e_L^{(+)}, e_T^{(+)}, e_{LT}$ are the corresponding ultimate strains. These failure criteria yield a rectangle and a parallelogram. They also assume in-plane stress and strain are independent for a structural analysis, but we should know this statement is not true. A typical example for that would be Moor's circle in Statics II. In fact, there are something called Tsai-Hill surface in the composite world, and this is a relatively accurate way to predict composite failure. The Tsai-Hill method does not create a circle, but similarly, it will generate an ellipse. Tsai-Hill method can be express as

$$\frac{\sigma_1^2}{s_L^2} - \frac{\sigma_1\sigma_2}{s_L^2} + \frac{\sigma_2^2}{s_T^2} + \frac{\tau_{12}^2}{s_{LT}^2} = 1. \tag{4.7}$$

The failure is avoided if the left-hand side of the equation is less than 1. In ANSYS ACP (Post), the failure contour follows the same standard that elements yield failures when results are larger than 1. However, Tsai-Hill method was proven by experiments that it is reasonably good for graphite/epoxy materials but yields an unconservative result for other type of composites. Therefore, there are total of nine types of failure criteria in ANSYS for you to select.

5. ANSYS

Presented below are two validation models, one with symmetric stacking orientation, the other with antisymmetric stacking orientation.

The validation described below is of a carbon fiber laminate under uniaxial tension as described in Examples 7.5 and 7.9 of Principles of Composite Material Mechanics by Ronald F. Gibson. This example was chosen as it will allow us to examine the stresses in each individual ply of a [45/-45/-45/45] or $[45/-45]_s$ laminate.

1. Open a new project in Workbench 19.1
2. Find ACP (Pre) in the Component Systems section, then create a standalone ACP (Pre) block in the project schematic.

Table 1. Material properties of AS/3501 carbon/epoxy.

Property	Value	
Young's Modulus along fiber direction – E_x	138	GPa
Young's Modulus transverse to fibers – E_y, E_y	9	GPa
Shear modulus – G_{xy}, G_{xz}	6.9	GPa
Shear modulus – G_{yz}	3.214	GPa
Poisson's ratio – v_{xy}, v_{xz}	.3	
Poisson's ratio – v_{yz}	.4	

3. Add a new material called AS/3501 Carbon Epoxy in Engineering Data. Double click **Orthotropic Elasticity** (under Linear Elastic) and type in values from the Table 1.
4. Double click Orthotropic Stress Limits and Orthotropic Strain Limits under the strength section and input corresponding values from Table 2 and 3.

Table 2. Orthotropic Stress Limits.

Property	Value (MPa)
Tensile X direction	1448
Tensile Y direction	48.3
Tensile Z direction	48.3
Compressive X direction	-1172
Compressive Y direction	-248
Compressive Z direction	-248
Shear XY	62.1
Shear YZ	62.1
Shear XZ	62.1

Table 3. Orthotropic Strain Limits.

Property	Value

Tensile X direction	0.01043
Tensile Y direction	0.00537
Tensile Z direction	0.00537
Compressive X direction	-0.00844
Compressive Y direction	-0.02756
Compressive Z direction	-0.02756
Shear XY	0.009
Shear YZ	0.009
Shear XZ	0.009

In addition to defining the properties of the composite, the ply type must also be defined. ANSYS will not warn you that you missed this step until after you have solved your model and are trying to post-process the results.

5. Double click **Ply Type** from the physical properties section onto the AS/3501 Carbon Epoxy material.
6. Expand the Ply type and make sure it is set to Regular.
7. Close Engineering Data.
8. Right click **Geometry --> Edit Geometry in Design Modeler**
9. Change units to **centimeter.**
10. Click on **XYPlane** in the design tree, then draw a 10 cm x 10 cm rectangle.
11. Click **Concept --> Surfaces from Sketches.** Make sure to confirm the sketch as your geometry. Click **Generate** to complete your surface.
12. Open Mechanical and assign the AS/3501 Carbon Epoxy Material to the model. Also assign a thickness of 1 mm. This is just a dummy thickness as the actual thickness will be set in ACP (Pre).
13. Apply a face sizing of **5 mm** and mesh the part. Close Mechanical.
14. Double click on Setup in the Project Schematic to open ACP (Pre).

ACP (Pre) is the preprocessing portion of the ANSYS Composites Package. The window should look much like the one below.

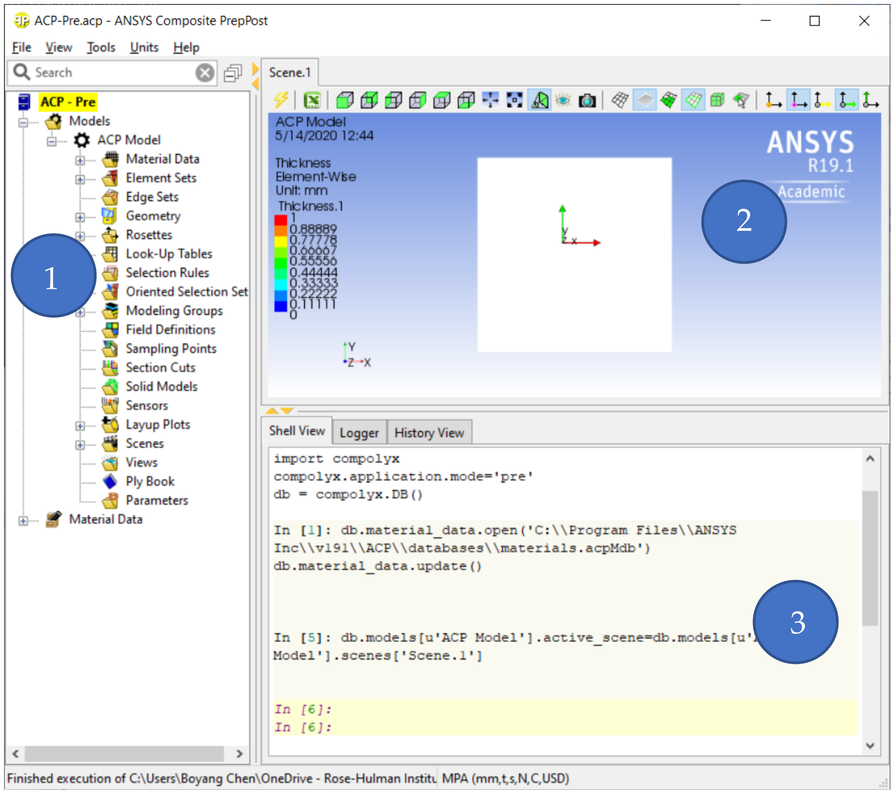

Figure 2. Capturing the ACP (Pre) window.

The window displays the model tree (1), the display window (2), and the command window (3). The command window is an area where you can use ANSYS classic commands. If the model is built using the model tree, the corresponding commands will be displayed in the command window.

ACP (Pre) is where you define every part of your composite layup. This includes defining the fabrics you will use to build your piece, the element sets that represent those fabrics, the reference fiber direction (rosettes), layup direction (oriented selection sets), and composite plies. There are other features of ACP (Pre) that allow you to build more complex models, like stack ups and section cuts, but they are out of the scope of this simple validation model.

First, we must define the fabrics that we will use to build our composite. We have already defined a material in Engineering Data, but this only provides the properties of the fabric, not the thickness.

15. Change the units to mm.
16. Expand Material Data in the model tree, then right click on Fabrics. Select **Create Fabric**.
17. In the Fabric Properties window, name the fabric *AS/3501 Carbon Epoxy.*
18. Select the predefined AS/3501 Carbon Epoxy material in the Material drop-down menu.

19. Define the material thickness as 0.25 mm, then click apply and OK. This thickness is the actual thickness of one layer of fabric. It will replace the dummy value of 1 mm that we input as the thickness in Mechanical in step 8.

Middle offset, essentially, allows you to create a mesh on the midplane of your composite and then translate this mesh through your plies. This then applies the mesh, boundary conditions, and forcing throughout the thickness of your composite. If this middle offset is left turned off (unfortunately the default), the boundary conditions and forcing are applied to only the midplane of the composite and your stress field varies throughout the thickness of your plies.

20. Right click on Element Sets and select **Create Element Set**.
21. On the display window, start from a point outside of the model, then slick and drag to select all elements. Now you should have something similar to Figure 3.
22. Select the check box for **Middle offset**.

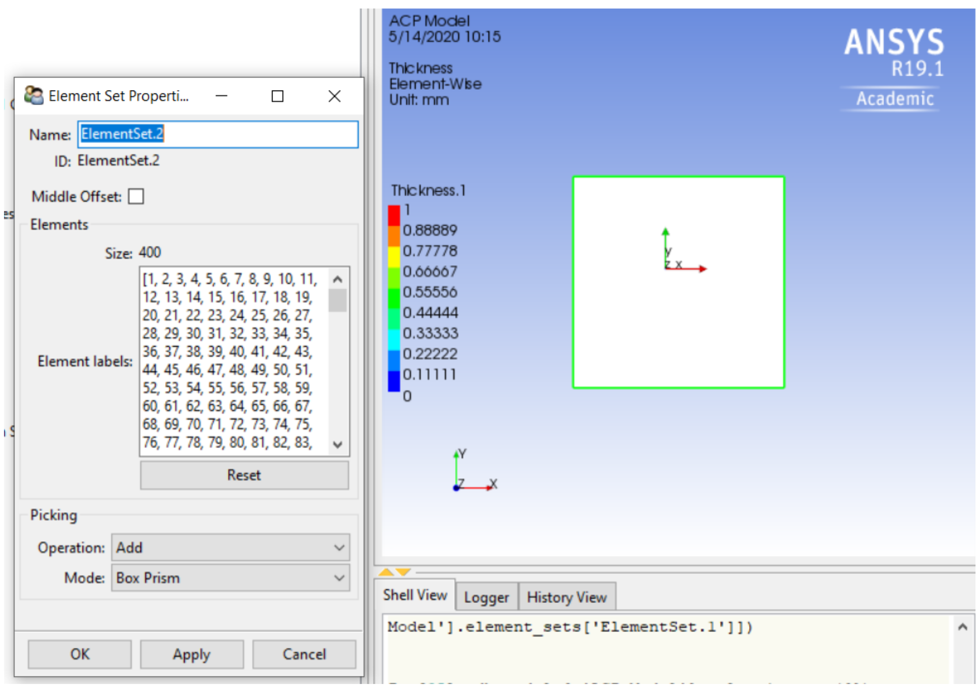

Figure 3. Creating a new Element Set.

For complex geometry, you should create one element set for each part of your design, as well as the Rosette and the Orientation Set that we are going to discuss below. Next, we must define a Rosette. A Rosette represents the axis from which the fiber angle is measured (the 0° line).

23. Right click on Rosettes and click **Create Rosette**.
24. Name the Rosette Rosie.1.

25. Click in the box labeled Origin and then click anywhere on your part in the display window.
26. Make sure Direction 1 is (1.0000, 0.0000, 0.0000) and Direction 2 is (0.0000, 1.0000, 0.0000) and the type is Parallel. (There are types of Rosettes for round, cylindrical, and spherical parts as well. Parallel Rosettes use Cartesian Coordinates for definition)
27. Click **Apply** and then **OK**.

Once the reference fiber direction has been defined, the direction in which the plies will be applied needs to be determined. (The direction the bottom ply to the top ply.) This is done by defining an oriented selection set. Multiple oriented selection sets can be created allowing you to lay certain plies in one area of your composite and other plies in another. Oriented Selection sets are defined using an Element set, a Rosette, and an orientation direction.

28. Right click on **Oriented Selection Sets** in the Model Tree and click **Create Oriented Selection Set**.
29. Put your cursor in the *Element Sets* box and **select All Elements** in the Element Sets section of the Model Tree. This will apply the Oriented Selection Set to all of the elements in the model. Any plies added on this Oriented Selection Set will cover the entire model.
30. Set the direction to **(0.0000, 0.0000, 1.0000)**. For our model, this will apply the first ply and then any subsequent ply will be laid up on top of it (the positive z-direction).
31. Under *Reference Direction,* **set the Selection Method to Minimum Angle**. The selection method defines how the reference direction should be defined when multiple rosettes are selected. Since we only have one Rosette, this does not affect our model. There are other options that you should read about before using.
32. Place your cursor in the Rosettes box and then **select Rosie.1** from the Rosettes section of the Model Tree.
33. Click Apply, then OK**.**

Now that we have defined our fabrics, our reference direction, and our layup direction, we can add the fabric plies to our model. There are two ways to create a laminate. For our validation model here, either one is not complicated, but if you need to create multiple plies for different structures, using **Stackups** as a module will be much more convenient than creating plies one by one.

Create laminate ply by ply:

34. Right click on Modeling Groups in the Model Tree and select Create Modeling Group.
35. Name the modeling group anything you want. This option is helpful for more complex parts that have varying ply sequences on different Oriented Selection Sets.

36. Now right click on the Modeling Group you created and select Create Ply.
37. Click in the Oriented Selection Sets box and select the Oriented Selection Set we created above.
38. Add the ply material AS/3501 Carbon Epoxy.1
39. If you remember, the composite we wish to create is a $[45/-45]_s$ composite. The way we defined our Oriented Selection Set determines the order that the plies need to be added. Because we are starting at the bottom ply and adding the other plies on top, set the Ply Angle of your first ply to 45.0. Click Apply and OK.
40. Repeat this process for the other three plies. Plies 2 and 3 should have an angle of -45.0. Ply 4 should have an angle of 45.0.

You may notice that your modeling group has three sublayers of plies like the following.

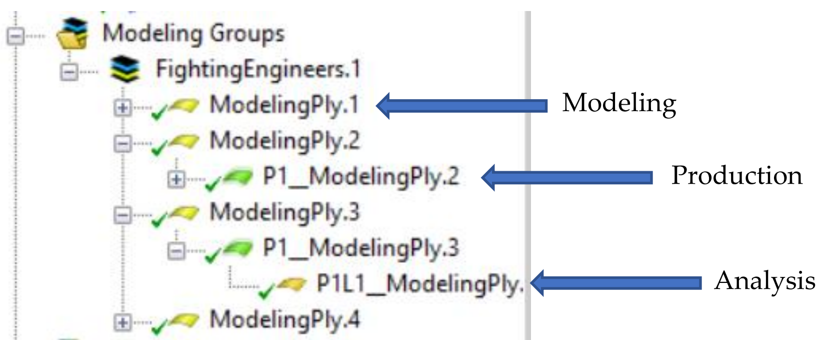

Figure 4. Showing ply structure in ACP (Pre).

The highest level is the Modeling Ply. This is where ANSYS stores the fabric selection, oriented selection sets, and draping (a really cool thing that is very much out of the scope of this tutorial). The second level is the Production Ply. This contains information for manufacturing such as when the ply is comprised of a manufactured stack up. The lowest level is the Analysis Ply. This ply is used in ACP (Post) for post-processing where you can see the stress and failure analysis later.

Create laminate in Stackups:

41. Right click on **Stackups** under Material Data and select **Create Stackup**.
42. In General tab, you will see two options for Symmetry and Layup Sequence. Select **No Symmetry** and **Top-Down**, respectively. Surely this easiest way is to select Even Symmetry and create only two plies in the table below.
43. Click the table under Fabric and select the *AS/3501 Carbon Epoxy.1* fabric you just created and have angle of 45 degrees.
44. Do the same for ply 2 to 4 with angles of -45, -45 and 45 degrees. You should expect the Stackup Properties look like Figure 5.

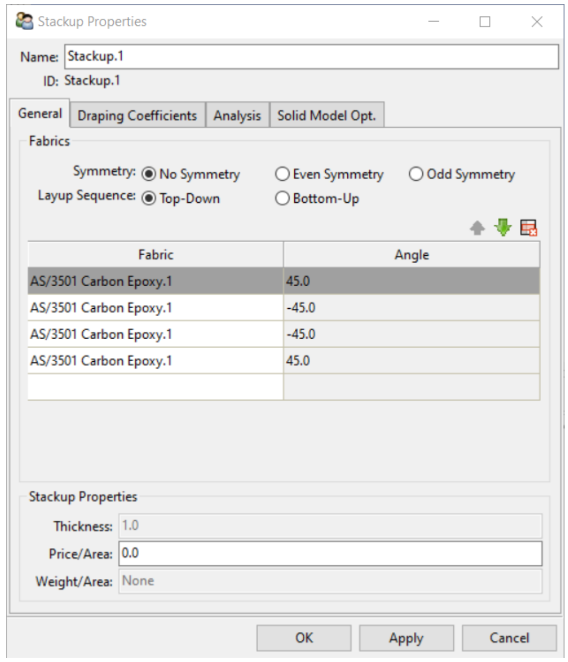

Figure 5. Creating a new Stackup Sequence for the symmetric laminate.

45. Click **Apply** and **OK**.

We have now finished the layup definition for our model. However, it is always not a bad idea to double check our Stacking sequence and layup direction.

46. Click on the first ply in Modeling Group. 1. On the tool bar of the display window, click the pink and light green arrow to show orientation and fiber direction. The angle starts from 0 at positive x direction and goes in a CCW direction. The orientation direction shows the stacking direction. You should also see the thickness of your laminate on the left side. the Figure 6 illustrates what you should be seen in the display window.

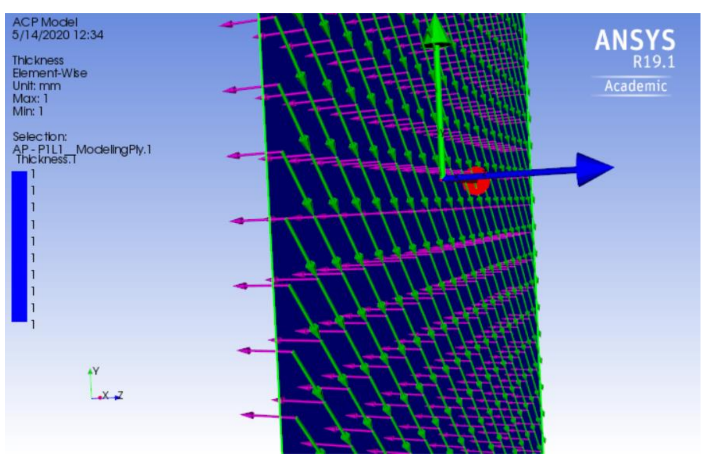

Figure 6. Capturing the display window when orientation and fiber direction indication are activated.

47. **Close ACP (Pre).**
48. **Drag and drop a standalone Static Structural** Analysis onto the project schematic.
49. Connect the Setup cell of the ACP (Pre) block to the Model cell of the Static Structural block. Select *Transfer Shell Composite Data* and refresh the Project.
50. Double click on Model to open Mechanical.

There are many ways to constrain the model. The one shown below is a recommendation for a simple geometry.

51. Apply a Displacement constrain on the left edge and constrain the **x component**.
52. Apply a Displacement constrain on the bottom edge and constrain **y and z components**.
53. Apply a force of 5000 N in the x direction to the right edge of the part.

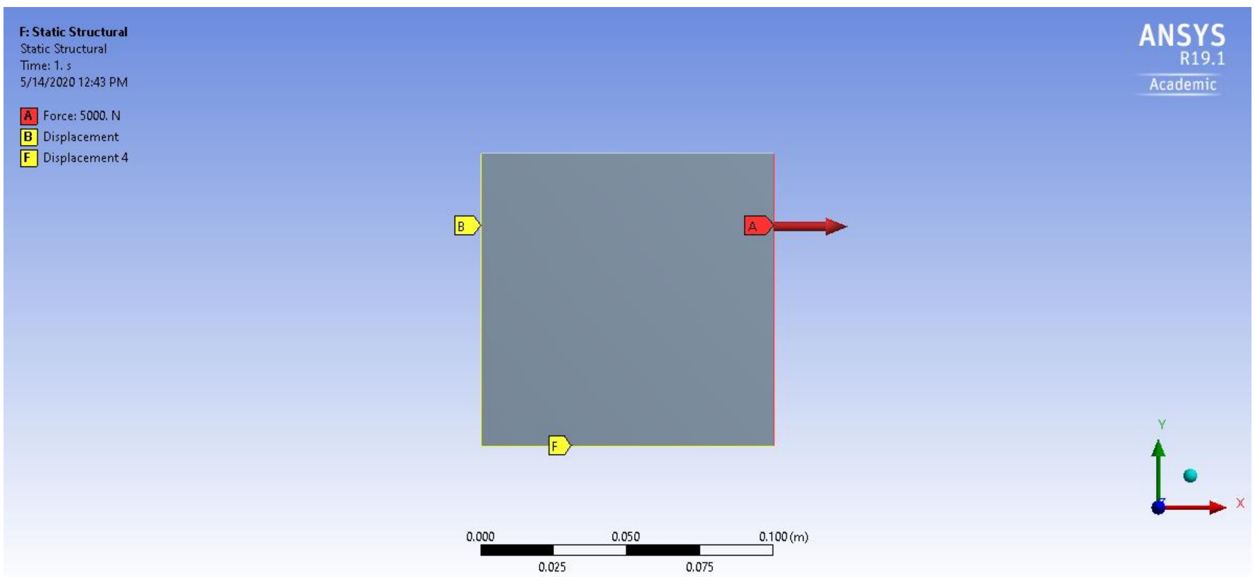

Figure 7. Showing constraints and the force on the 2D model

From this point on, you can solve the model and go to ACP (Post) to check the stress analysis result, but you can view stress and strain in Mechanical to give you model a sanity check. This may not be the most efficient way to view stress analysis of your design, but by checking one or two plies of your structure, it will save you a lot of time if you choose to go back and forth between Mechanical and ACP (Post).

54. Click Solution and select Normal Stress in the Stress tab. Select ply 1 in Imported Plies in Outline tree for Normal Stress. Another way to create a Normal Stress is to right click the ply in Imported Plies and select **Insert Environment --> Static Structure --> Normal Stress**.
55. Select **Orientation** to be **X Axis**. For a symmetry laminate, position does not matter for our planer stress analysis, top, middle, and bottom should have the same result.

56. Solve the model and close Mechanical. You should have a uniform result of 50 MPa for the first ply. One of the biggest advantages of showing solution in Mechanical is Mechanical will show stress results in term of the global coordinates.

Now that the model has been constrained and solved, we can post process the results to determine the stresses in the individual plies of the composite, as well as evaluate various failure criteria.

57. Drag and drop an ACP (Post) block onto the Model cell of your ACP (Pre). Connect the Solution cell of the Static Structural to the results cell on the ACP (Post).
58. Update the Project to share the results between programs. Double click on Results to open ACP (Post). It should look identical to the ACP (Pre) window from earlier. There should now be an object called Solution.1 under the Solutions file in the Model Tree.
59. Right click on Solution.1 and Create Stress.
60. We wish to see the stresses in the x-direction, so set component to s1. Also set the spot to mid. This setting allows us to see the stresses at the midplane of the ply. You can examine interlaminar stresses by using the bot and top options for spot. Additionally, make sure the check box next to Ply wise is checked. Click apply and close the window.
61. Right click on Solution.1 again and Create Deformation.
62. In the Deformation window, set the component to x. This will allow us to see the strains in the x direction only. The default deformation is the root sum squares of the x, y, and z components of the deformation.

Only one solution type can be viewed at any given time (stress, deformation, failure, etc.). You will have to right click on the type of plot you want to analyze and click show. The deformation plot will display on the screen automatically, while the stress or failure plots will stay white. This is because the deformation plot is global, and as such, it does not require another selection. For ply wise plots, like stress and failure, you will have to select the desired Analysis Ply from the Model Tree.

The Failure criteria require an extra step before you create the failure solution. We must first define the criteria we wish to see, Tsai-Hill for this case. Tsai-Hill failure criterion like we discussed above in section 4, anything bigger than 1 is a failure.

63. Right click on **Definitions** in the Model Tree and select **Create Failure Criteria**
64. There are nine different criteria for Reinforced Plies. Composites are inherently difficult to analyze and therefore many different failure criteria exist for different situations. Select **Tsai-Hill**. Click apply and close the window.

You may have noticed the Sandwich Criteria. Many composites are created by wrapping a sandwich material in a reinforcing fabric. This allows you to create composites with higher moments of inertia (more material away from the bending axis) without stacking tens or hundreds of layers of material. The sandwich core is typically lightweight with very low strength of its own. If you wish to analyze these failure criteria in the future, make sure to do some research and understand what they mean.

65. Now right click on **Solution.1** and **Create Failure**
66. Select **Ply-Wise** and set the Failure Criteria Definition to FailureCriteria.1. This tells the program to only show the failure criteria that you defined in the previous step.

The Show Critical Failure Mode box is already checked. This means that ANSYS will analyze every element and show the failure criterion that is closest to failure if you select more than one. You can check Show Critical Layer if you do not check ply wise. This will enable you to see which layer will fail for each element.

67. Click apply and close the window.

You can now show the failure plots and analyze them by selecting the Analysis Plies.

Now that we have Post-Processed for all of our results, we can verify them with the analytic solution. As shown in example 7.9 of Principles of Composite Material Mechanics, we should expect to get the following stresses in each ply shown in Table 4.

Table 4. Stress distribution for the symmetric laminate.

Location	σ_x(MPa)	σ_y(MPa)	τ_{xy}(MPa)
Ply #1	50	0	21.2
Ply #2	50	0	-21.2
Ply #3	50	0	-21.2
Ply #4	50	0	21.2

68. Right click on the stress plot you created under solutions and open the Properties window.
69. Because we applied our boundary conditions and forces to entire faces of the part, the stress distribution should be the same at the top, mid-surface, and bottom of each ply. For this reason, we can leave Spot set to bot to get our desired results. Repeat the following process for mid and top to prove to yourself that this is true.
70. Set component to **s1** and select Apply and OK.
71. Make sure that the stress plot is actually selected by right clicking on Stress and clicking Show.
72. Now select the each of the modeling plies one at a time and look at the stress contours in the display window.

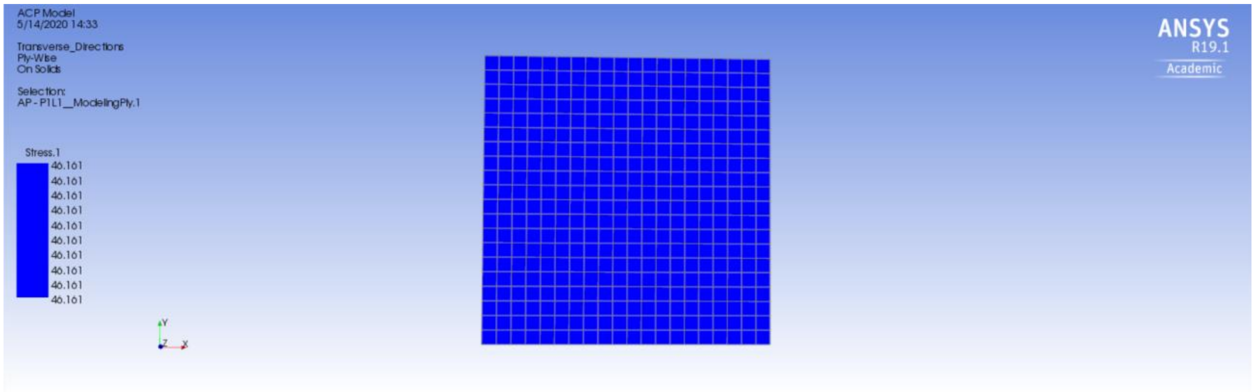

Figure 8. Capturing stress contour for the s1 direction of Ply 1.

Check the shear stress and the other axial stress by viewing the **s12 and s2 components** in the stress plot, respectively. You may notice that the s12 direction has a shear stress of -25 MPa for plies one and four and 25 MPa for plies two and three. The s2 direction has a stress of 3.8386 MPa for all four plies. When we constrained the model, we only added displacement constraints in the y-direction to the center of the part. The part is free to deform in the y-direction due to Poisson's ratio. So why are we seeing a stress at all?

ANSYS allows you to see the stress contours in the 1-2-3 coordinate axes. These axes are commonly used to analyze composites, but they are not synonymous to the x-y-z coordinates you are familiar with. The 1-2-3 principle coordinate axes are a local coordinate system that corresponds to the longitudinal and transverse fiber directions of the composite. As you can see in the following image, the coordinates are offset by the fiber angle in the plane of the fabric. The 3-direction in local coordinates and z-direction in global cartesian coordinates directly correspond because we are looking at generally orthotropic lamina. If you were to check the s3 stress contour in ACP (Post) you should see 0 MPa of stress in all the plies.

After applying equation 1.2 and 1.3 to the stresses obtained in ACP (Post), we obtain the following stresses in the plies shown in Table 5.

Table 5. Stress distribution for the symmetric laminate.

Location	σ_x(MPa)	σ_y(MPa)	τ_{xy}(MPa)
Ply #1	49.9998	-0.0002	21.1612
Ply #2	49.9998	-0.0002	-21.1612
Ply #3	49.9998	-0.0002	-21.1612
Ply #4	49.9998	-0.0002	21.1612

The error in these values compared to the theoretical values are within 0.2%.

We have now validated the case of uniaxial tension of a composite with non-zero fiber angles for a symmetric stacking sequence laminate. Next, we will discuss the procedure for an antisymmetric stacking sequence laminate.

73. Double click Geometry and use **Split Edge** on the **left edge** to split into two edges.

In step 44, we create a stackup with the fiber direction as $[45/\text{-}45]_s$, this time we need to change the fiber direction for the third and fourth ply to 45 and -45 degrees as Figure 9 shown.

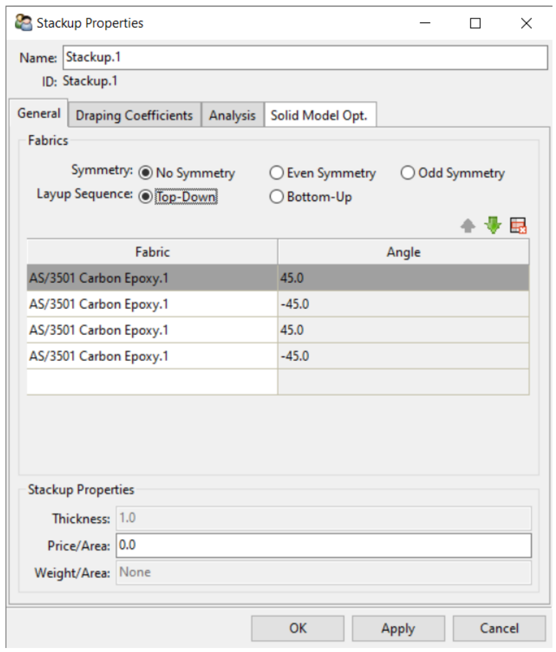

Figure 9. Creating a new Stackup Sequence for the antisymmetric laminate.

74. Close Geometry and open ACP (Pre).
75. Find Stackup.1 you created and change the angle of the last two plies.
76. Close ACP (Pre) and right click Model of the Static Structure block and update.
77. Double click Model to open Mechanical.
78. Apply **Fixed Rotation** at the **middle of the left edge** on x axis.
79. Create four Normal Stresses in Solution.
80. Assign ply 1 to 4 for these Normal Stresses in x direction, and set the position to Top/Bottom, so you can see two stresses at once.
81. Click Solve and you should have the result in Figure 10.

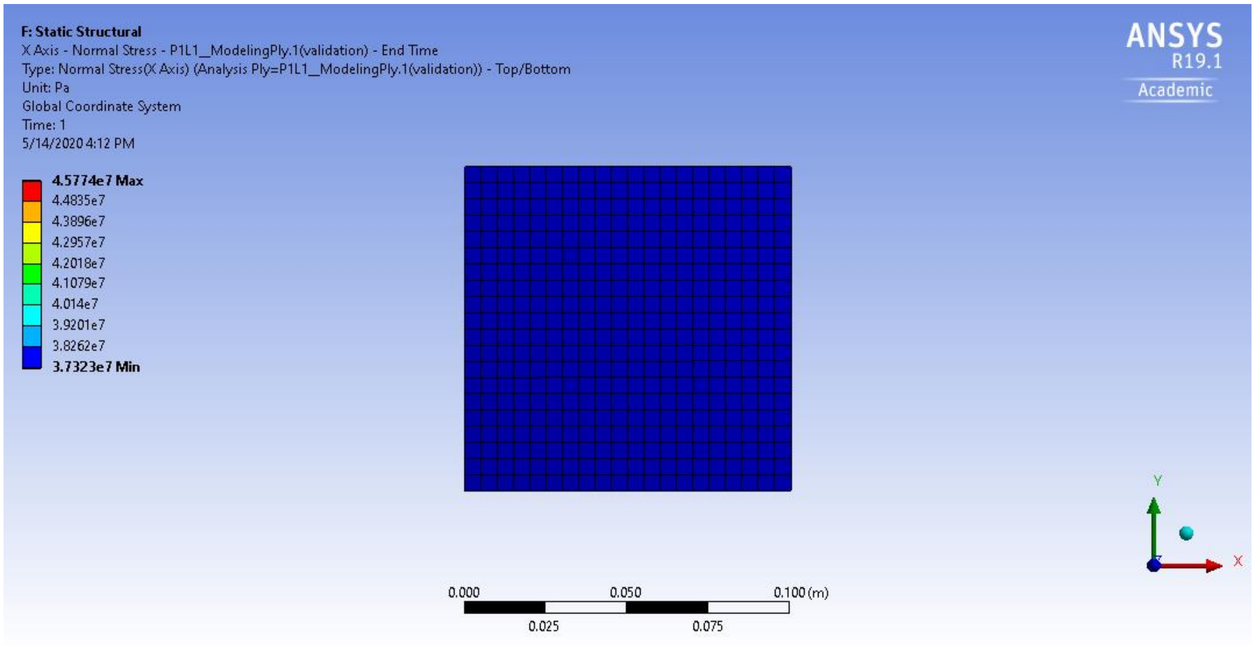

Figure 10. Capturing stress contour for the x direction of Ply 1.

If you rotate each ply, you should see there is a rotation resultant for the uniaxial load we applied on the right edge. This is because the laminate is antisymmetric and [B] matrix in equation 3.4 is no longer zero. This is also the reason we need to define the stress result separately for top and bottom of each ply. You should have the same result as the example 7.10 in Principles of Composite Material Mechanics by Ronald F. Gibson in Table 6.

Table 6. Stress distribution for the antisymmetric laminate.

Location	σ_x(MPa)	σ_y(MPa)	τ_{xy}(MPa)
#1 Top	37.3	-12.7	-6.2
#1 Bottom	45.8	-4.2	-15.5
#2 Top	62.7	12.7	34.0
#2 Bottom	54.2	4.2	24.7
#3 Top	54.2	4.2	-24.7
#3 Bottom	62.7	12.7	-34.0
#4 Top	45.8	-4.2	15.5
#4 Bottom	37.3	-12.7	6.2

To build a more comprehensive understanding of composite, peruse the book Principles of Composite Material Mechanics or take Dr. Riley's Mechanics of Composites course.