Numerical Analysis of Composite Coupons With Complex Laminate Schedules Under Tensile Loading

Submitted by: Kosuke Ishikawa
Submitted to: Dr. James Locke
Date Submitted: August 2nd, 2004

Acknowledgements

I thank Dr. James Locke, my supervisor of the directed project, for his support and help to complete the directed project and numerous advices over the past two years and seven months of my study at Wichita State University.

1 Abstract

Wichita State University is planning to perform tensile testing on composite materials with various laminate schedules to determine their modulus and strengths. This study is to identify the optimum configurations using an ANSYS finite element model. The cases studied are focused on the effect of boundary conditions, laminate schedule and the aspect ratio of the specimens. Conducting such analyses allows for more effective experimental testing in real experiment.

The cases studied were balanced symmetric, biased symmetric, and biased unsymmetric laminate cases since they are the three unique cases from the proposed experimental test cases.

In order to quantify and measure average strains in the test specimens, the aspect ratio is recommended to be six or higher so that the strain values measured are affected by the boundary condition will be minimized and will not affect the values measured.

Strain values for all proposed test cases are listed in the Appendix C.

Table of Contents: Page

2 Objective

The goal of this project is to identify the effects of material layup, aspect ratio and boundary conditions on the predicted strain field for axially loaded composite specimens. These results will be used to design experimental test specimens that will be used to determine laminate stiffness and strength properties. It is desired to develop a laminate design that produces easily measured results for a wide range of material layup and boundary conditions.

3 Assumptions:
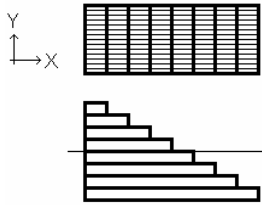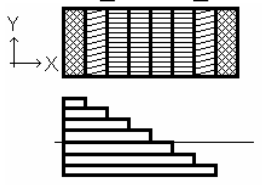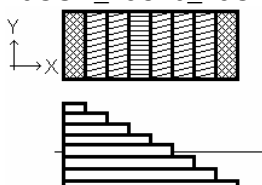
Since the studies were done by finite element analysis method, the values indicated here will not be the same as the results from the actual testing. These are the assumptions made for this research project.

- No temperature changes occur while the load is being applied
- Perfect Laminate
  - Distribution of the fibers and epoxy are ideal
  - No void in the specimens
  - The bonding between layers will not allow slipping
  - Epoxy is strong enough to withstand the load applied
  - No prestress exist in the coupon
- Adhesive used to bond the strain gage to the specimen is perfect and causes no shear or slip
- Small out of plane displacement
- Thin plate theory applies
- Materials have linear elastic behavior

4 Cases to Study and the Model Setup

4.1 Descriptions of the Cases

Out of the 21 cases proposed for the testing, three unique cases were chosen to study the effects of the applied boundary conditions and the aspect ratios. Although there is no case with identical laminate schedules, all cases can be classified into three groups; balanced symmetric, biased symmetric and biased unsymmetric. Case #1 represents a balanced symmetric case, while case #5 represents a biased symmetric case and case #9 is a biased unsymmetric case. The table below shows the laminate properties of the cases to be studied.

Case#	Configuration	Laminate Description
1	SC_GDU16	Glass at 0°, 0.0004075m (8) Total thickness: 0.0032600m
5	55CUD_30C20_15GDB	Biax at 0°, 0.000280m (1) Carbon at 20°, 0.0005183m (1) Carbon at 0°, 0.0005183m (4) Carbon at 20°, 0.0005183m (1) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m
9	10GUD_75C20_15GDB	Biax at 0°, 0.000280m (1) Carbon at 20°, 0.0005183m (2) Glass at 0°, 0.0004075m (1) Carbon at 20°, 0.0005183m (3) Biax at 0°, 0.000280m (1) Total thickness: 0.0035590m

2

4.2 Model Setup on ANSYS

The models were made using ANSYS. The aspect ratio of the element used is fixed at 1: 1, and the size of one element is 0.05m by 0.05m. The number of elements in the Y direction is set to be 12, while the number of the elements in the X direction varies from 24 to 120, depending on the aspect ratio of the specimen. The number of elements in each direction is enough that the results from a denser element mesh case differ only slightly regardless of the schedule of the laminate. From the settings noted above, the width of the coupon is set to be 0.6m and the length is 1.2m for the L/W=2 case and length is 4.8m for the L/W=8 case.
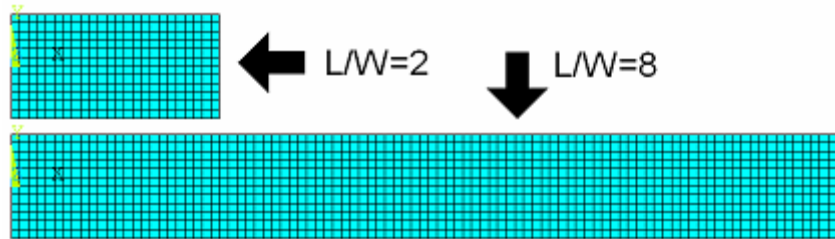
Finite element mesh geometry

The applied load is 1200N, which is equivalent to 2000N/m of load for the classical lamination theory study. There are several applied boundary conditions which will be discussed in the later section.

5 Study 1: Effect of Material Layup

5.1 Overview

This chapter will present the laminate properties for each testing case. The behavior of the specimen is dependent on the layup, materials, and laminate, including the layer thickness, and fiber direction. It is known that a biased layup will behave differently from a balanced layup. Also, the laminate with symmetric schedule will behave different that these with an unsymmetirc schedule.

5.2 Behaviors of Laminate Under Special Conditions
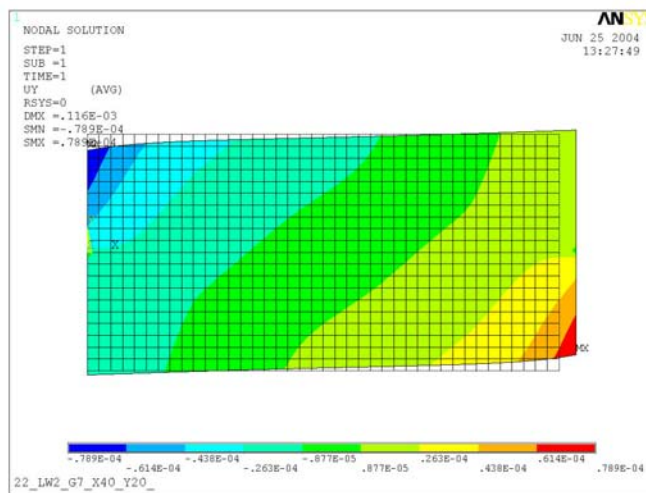
Biased Laminate

When a laminate is biased, the specimen will deform to one side when the load is applied

\
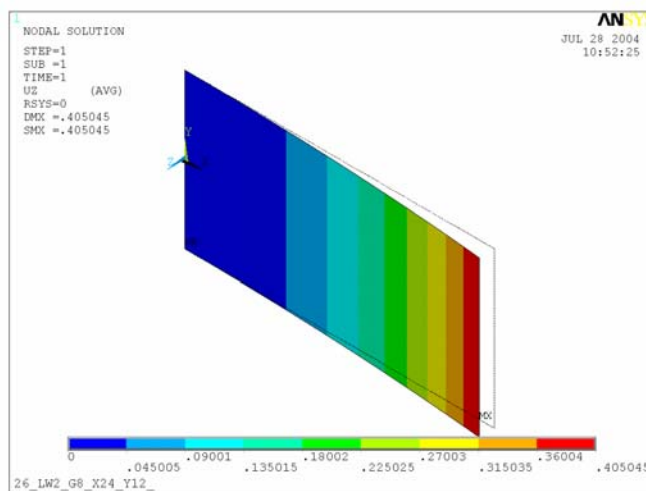
Unsymmetric Laminate

If the layup is not symmetric, the coupon have curvature, causing the laminate to bend in a vertical direction in the case the specimen has a free end.
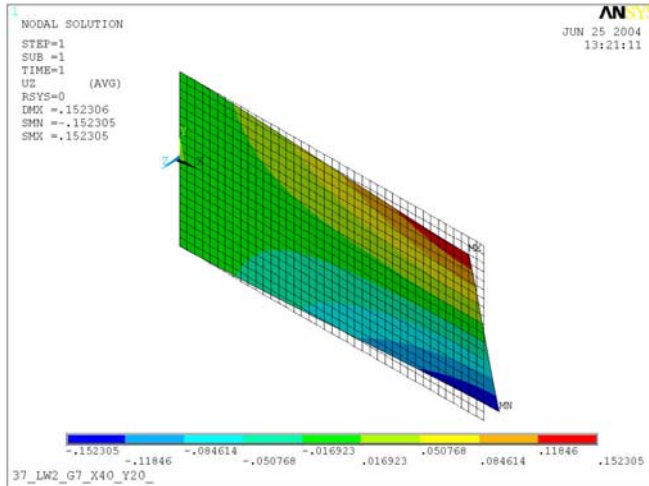
Laminate with Coupling

If the coupon has coupling, the specimen will be twisted as a load is applied. If only one end of the coupon is constrained, the twist will be fully developed.

5.3 Classifications of the Cases

Three cases, 1, 5 and 9, are chosen for this study and they will represent each group of unique laminate schedules exist in the entire cases.

Case#	Title	
1	SC_GDU16	This is a balanced, symmetric layup therefore the problem can be treated as 2-D plate
5	55CUD_30C20_15GDB	The laminate is symmetric but contains two layers at 20 degree offsets so the coupon will have deformation to the side.
9	10GUD_75C20_15GDB	The laminate is unsymmetric so the coupon will have curvature and also five layers contain 20 degree offsets so the coupon will also have deformation to the side.

5

5.4 Results and Summaries

The results shown below are from the L/W=2, BC #1.
Case 1: Uniaxial Case
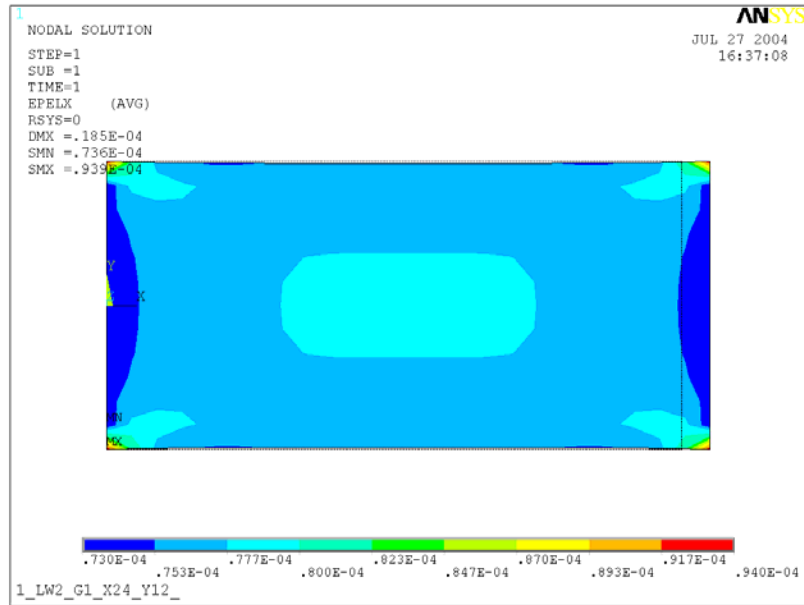
Axial Strain (εX)

Transverse Strain (εY)
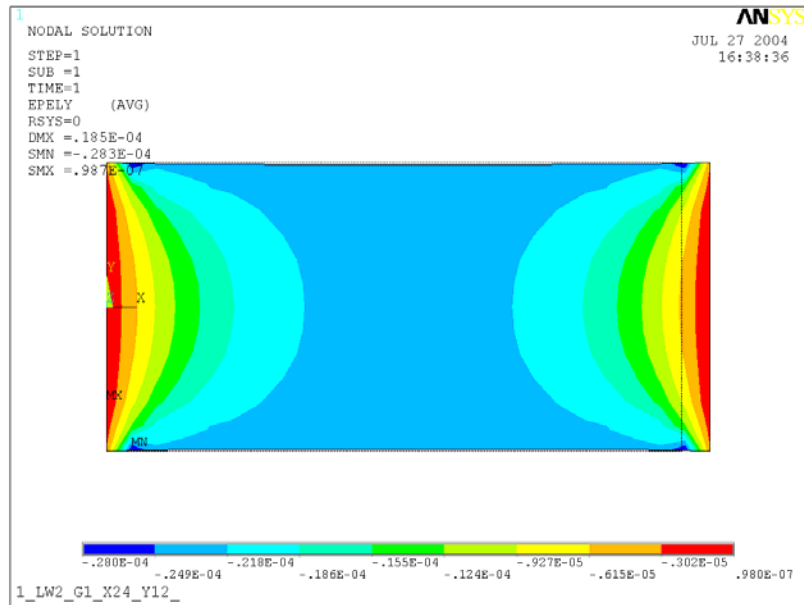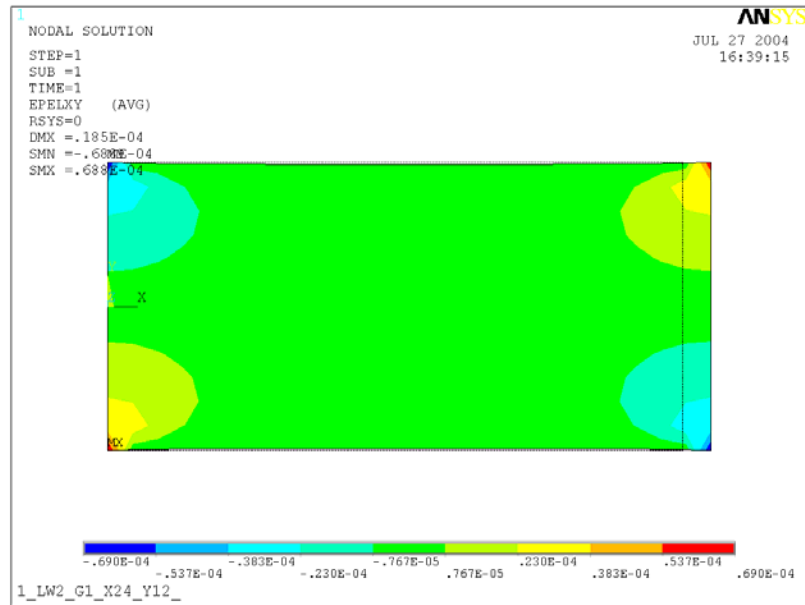
Shear Strain (γXY)

Summary of the plots:

By observing the plots, the axial strain values are stable in general but there are high strain places near the corners caused by the constraints. The transverse strain shows high values near the ends. The shear strain is near zero and generally uniform, but the all corners have high strain values due to the constraints applied.

Case 5: Biased, Symmetric Case
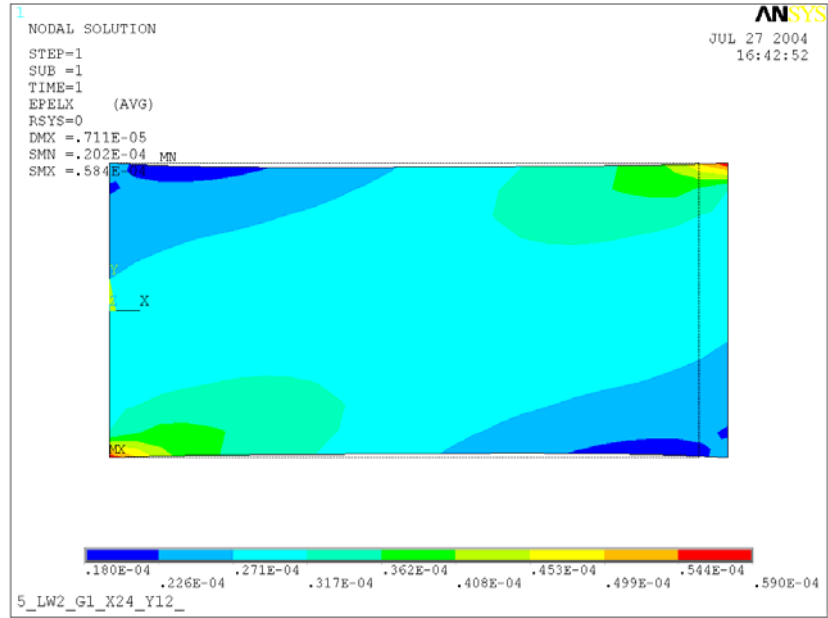
Axial Strain (εX)

Transverse Strain (εY)
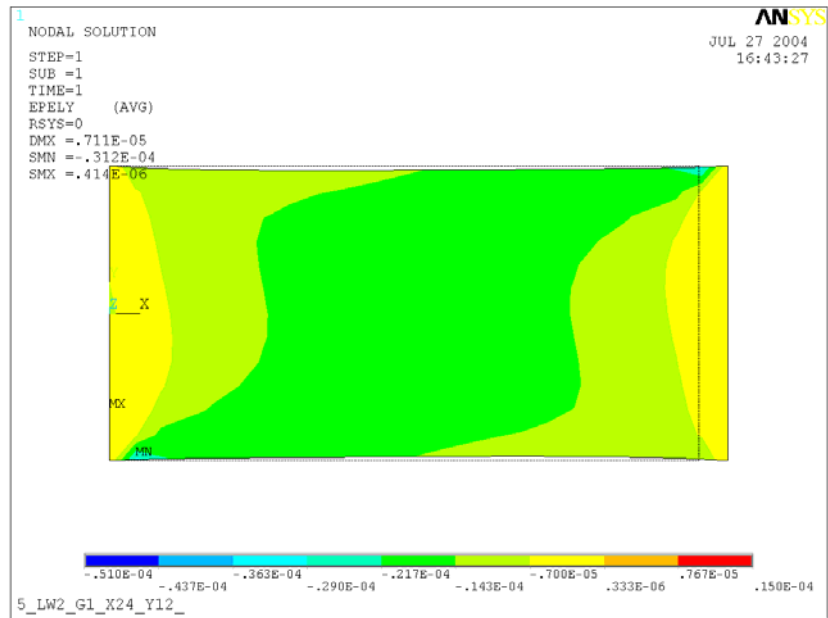
Shear Strain (γXY)
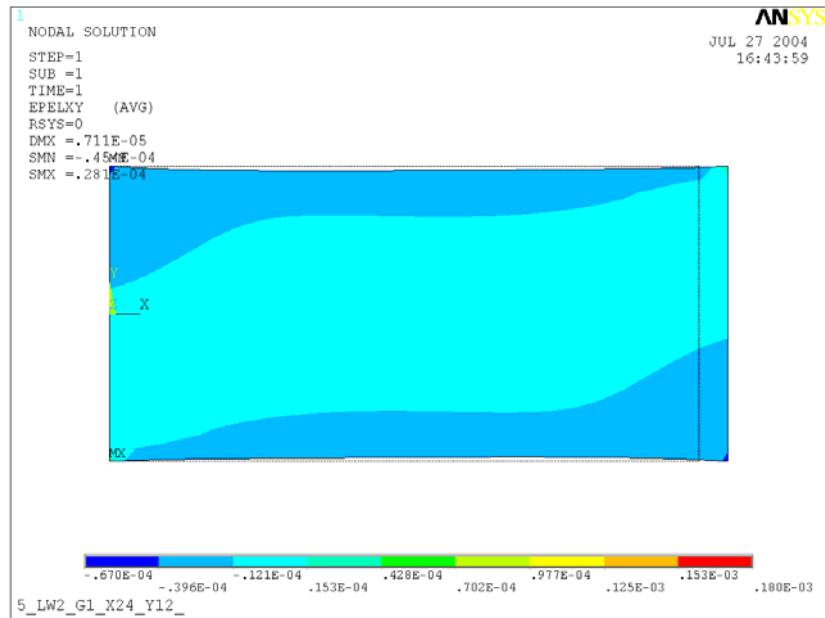
Summary of the plots:

Diagonal contour lines can be observed and they are caused by the offset of the fibers angle for the all strain components.
For axial strain, the upper left and the lower right corners are under compression therefore the values are lower than average, while part of the coupon at the lower left and the upper right corners shows high strain values. Overall the center portion of the model has small changes in the strain value at entire length. This should allow getting accurate average strain values by the gages.

Case 9: Biased, Unsymmetric Case
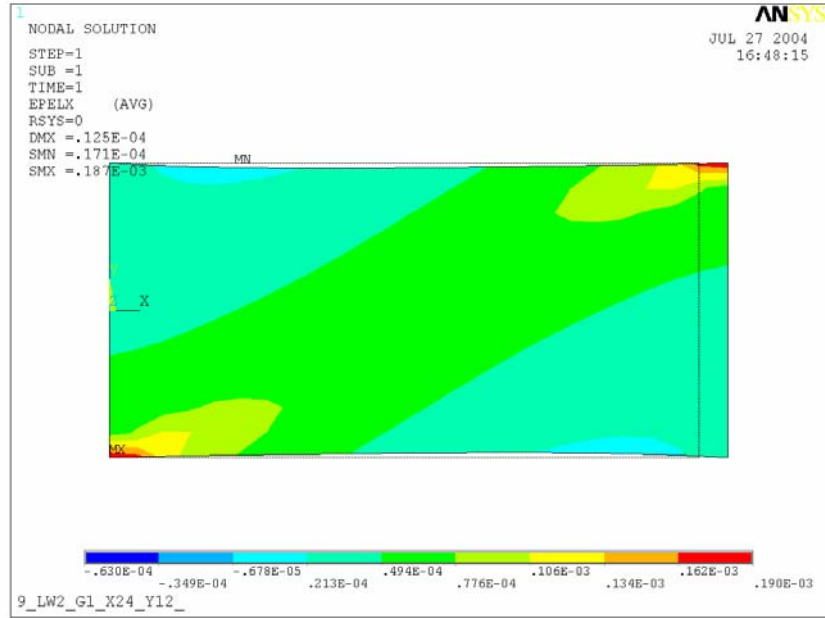
Axial Strain (εX)

Transverse Strain (εY)

Shear Strain (γXY)

NODAL SOLUTION
STEP=1
SUB =1
TIME=1
EPELXY   (AVG)
RSYS=0
DMX =.125E-04
SMN =-.258E-03
SMX =.117E-04

JUL 27 2004
16:49:14

-.260E-03   -.822E-04   .956E-04   .273E-03   .451E-03
      -.171E-03   .667E-05   .184E-03   .362E-03   .540E-03
9_LW2_G1_X24_Y12_
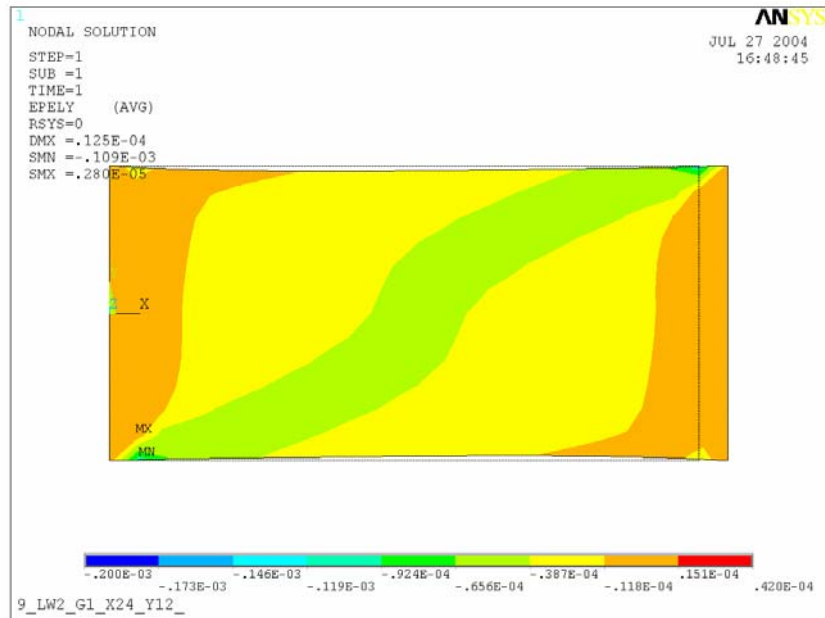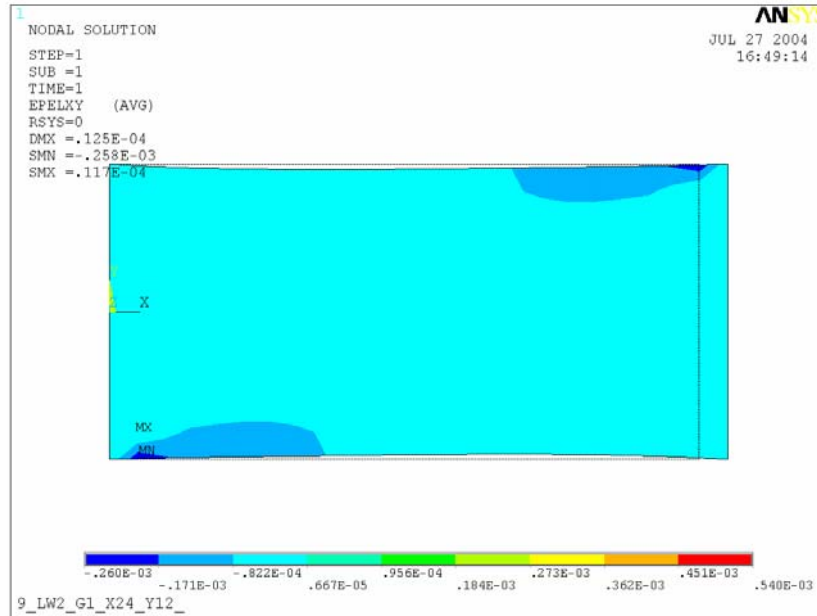
Summary of the plots:

Diagonal contour lines can be observed for the all strain components and they are caused by the offset of the fibers.

For axial strain, the upper left and the lower right corners are under compression therefore the values are lower than normal, while part of the coupon at the lower left and the upper right corners shows high strain values. Compared to the case with symmetric layup, the change in the strain value is more severe, meaning that the boundary condition affects more for the case with unsymmetric laminate schedule.

Axial, Transverse and Shear strains measured at the Y=0 for L/W=2, BC1
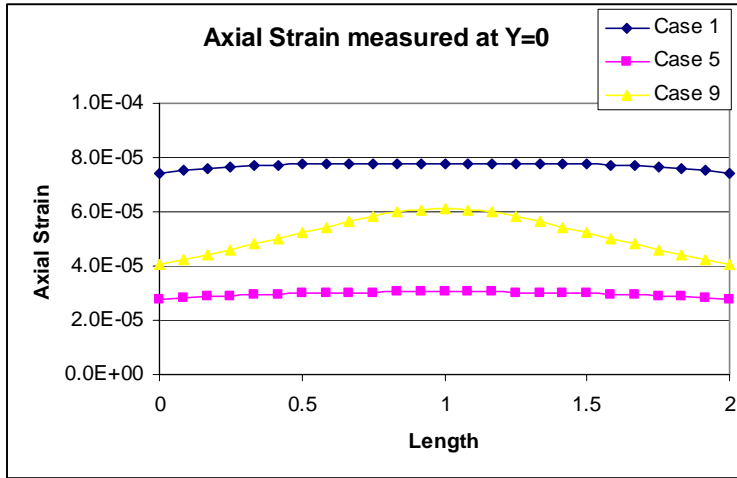
Axial Strain
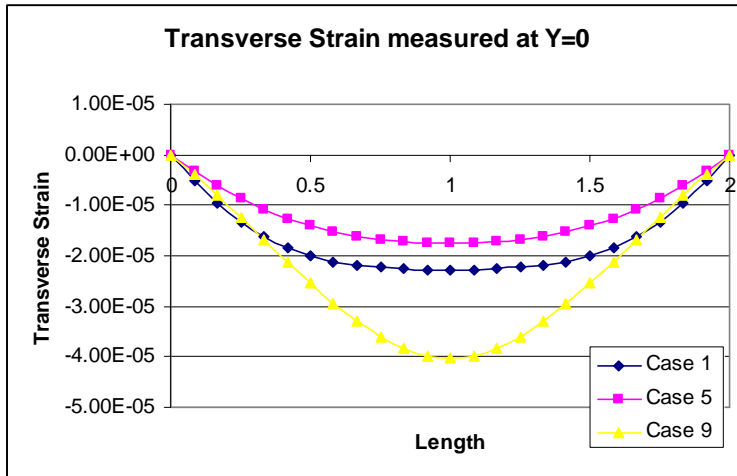
Transverse Strain
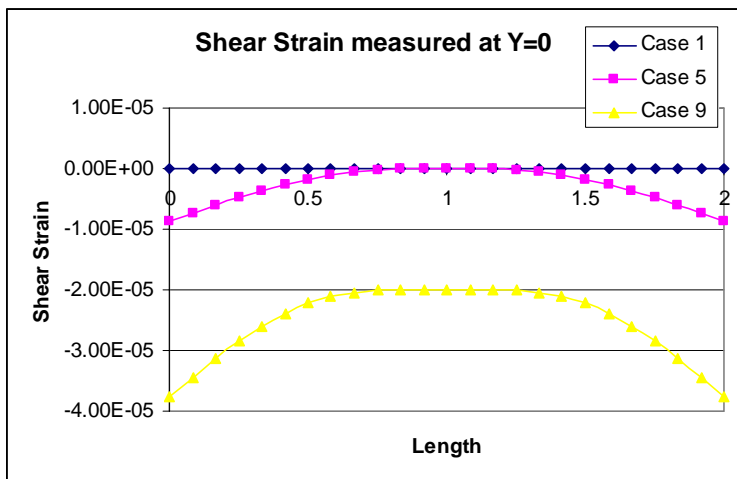
Shear Strain

Transition of the strain values along the center line

When the strains are measured at the center of the specimen, which is Y=0, there are some significant differences found by observing the trends of the curves. Since the strain is measured by a strain gage, which reads the only average value, the gradient of the value should ideally be zero.

For axial strain, the case 1 and 5 maintain uniform value at the entire length, but the case 9 which is unsymmetric case, varies its value. This could be a problem for placing a strain gage and must be observed if the same situation can occur in the other cases and also how the aspect ratio affects the result.

The transverse strain values are zero at the both ends for the all the cases due to the directional constraints applied at the center, the curves will be gradually decreases as the distance from the ends increases, in all cases the value will have no change at the middle, but the amount of change in value is largest for the biases unsymmetric case.

The shear strains are different for either balanced or biased cases. While the balances case has a single value along the length of the specimen, the biased cases have change in the value within the boundary. This can be explained since the biased cases have their directional modulus offset from the longitudinal axis. The value will be stable near the center of the boundary, where the strain gages usually be placed, so this will not likely to affect the result value.

Both symmetric cases have shear values close to zero near the center, but the unsymmetric case has non zero value even at the center, meaning that the shear strain cannot be eliminated for this case. Close attention must be paid for the cases with unsymmetric laminate.

6 Study 2: Effect of Applied Boundary Condition

6.1 Overview

This chapter will be discussing the relationship between the results for certain cases and the applied boundary conditions. Some laminates can be easily affected by the boundary condition, while others do not have much difference in the results for different configurations. In this study, there are two kinds of boundary conditions to be tested.

Boundary Condition #1
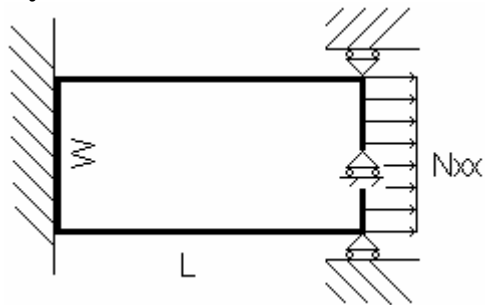
This boundary condition allows no movement for any directions at X = 0. At X = L, the constraints are set in similar ways, but displacement in the X direction is allowed.

Boundary Condition #2
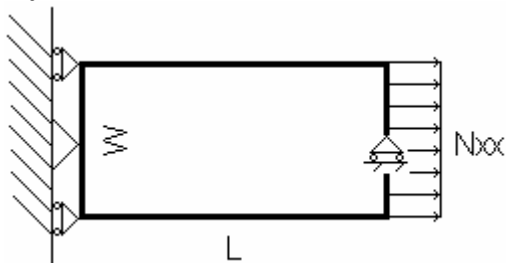
The other boundary condition still does not allow movement in the Y direction at both ends but this restriction is only applied at the center (Y = 0). The end at X = L is set so that the entire end moves the same amount in the X direction (UX is constant at X=L).

Note:

Both boundary conditions will not have displacement in the Z direction, UZ.

Since the cases with lower aspect ratios are largely affected L/W=2 is chosen for this case. The strain values were measured at the center of the coupon (X=L/2, Y=0).

6.2 Results and Summaries

The upper half of the figure shows the strain results under BC1, while the lower half shows result of BC2. The scale is matched for both boundary conditions therefore the colors within the figure matches.

Case 1: Uniaxial Case
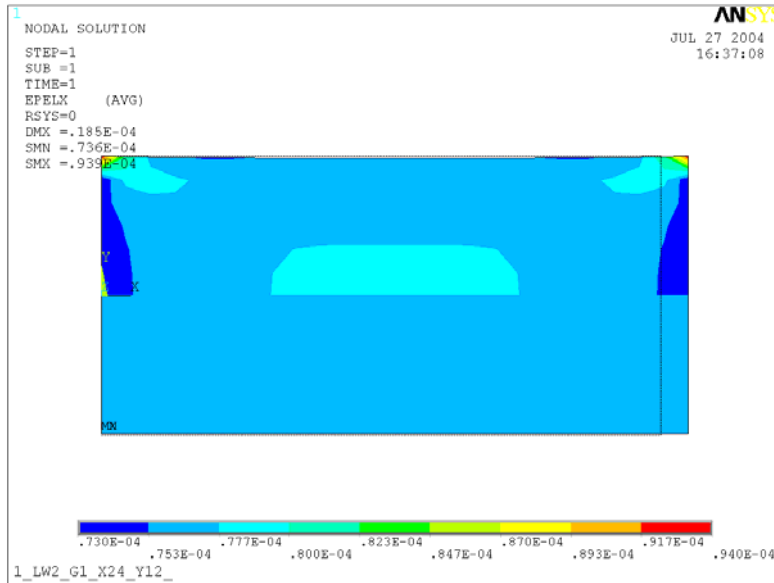
Axial Strain (εX)

Transverse Strain (εY)
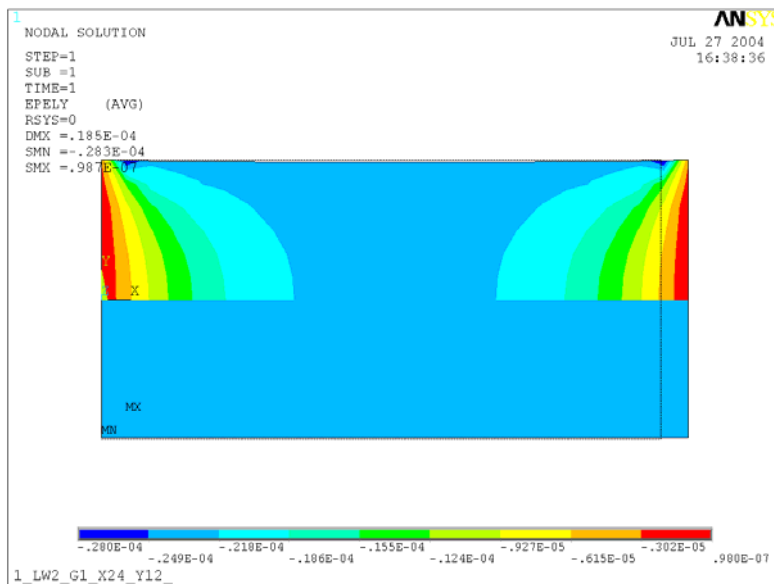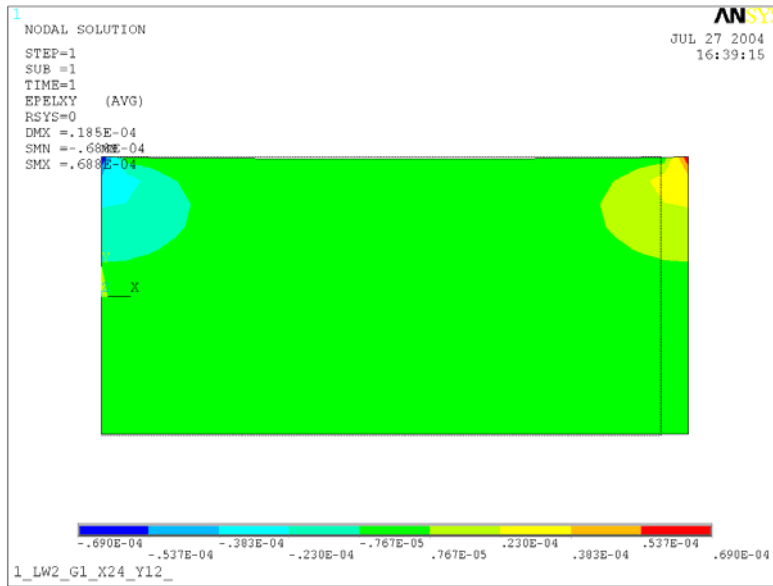
Shear Strain (γXY)

All the numerical values are taken at the center (X=L/2, Y=0).

	B.C.#1	B.C.#2
Axial Strain	7.788E-05	7.727E-05
Shear Strain	-7.295E-17	-1.034E-16
Transverse Strain	-2.284E-05	-2.241E-05

There are no large differences in strain values between the boundary conditions. Therefore, the laminate with uniaxial layer will not be affected by the boundary conditions.

Case 5: Biased, Symmetric Case
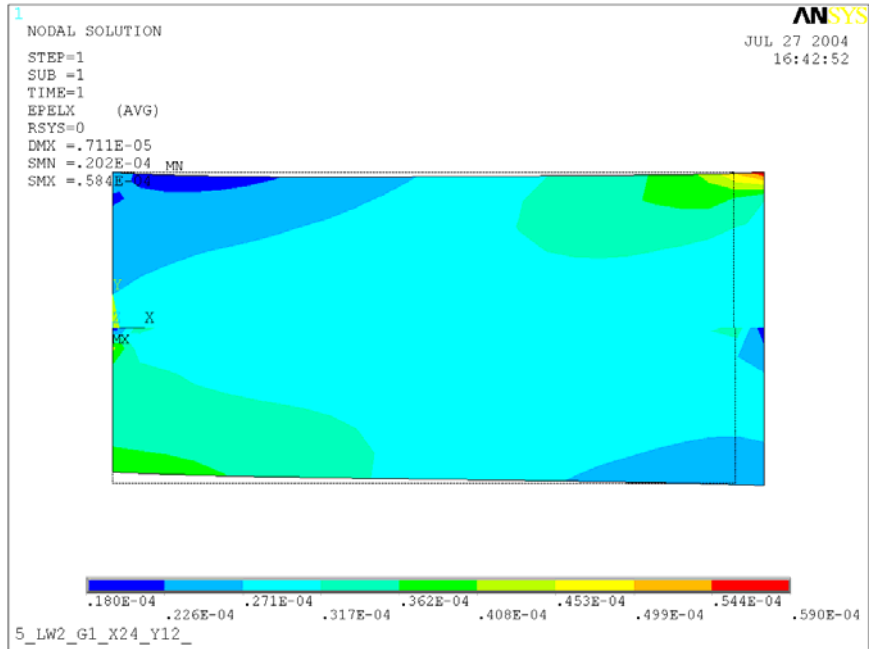
Axial Strain (εX)

Transverse Strain (εY)

Shear Strain (γXY)


```
1
NODAL SOLUTION                                              AN SYS
STEP=1                                                 JUL 27 2004
SUB =1                                                    16:43:59
TIME=1
EPELXY   (AVG)
RSYS=0
DMX =.711E-05
SMN =-.46ME-04
SMX =.281E-04
```

```
-.670E-04        -.121E-04        .428E-04        .977E-04        .153E-03
      -.396E-04        .153E-04        .702E-04        .125E-03        .180E-03
5_LW2_G1_X24_Y12_
```
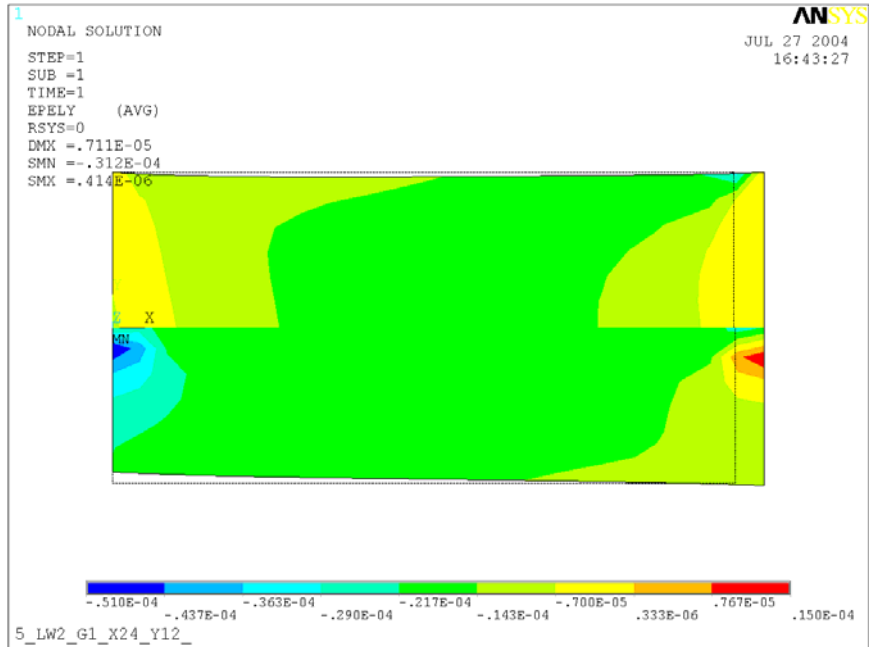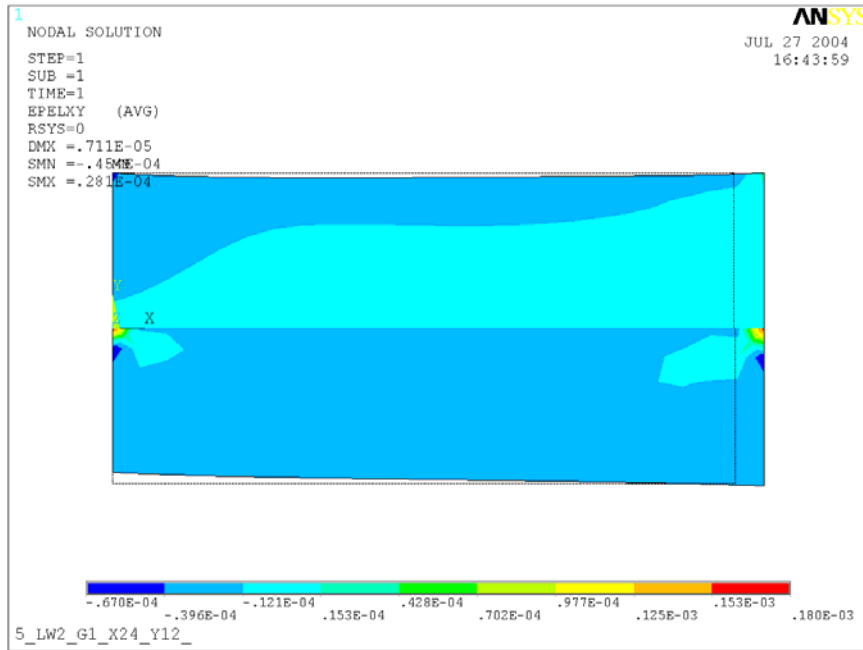
All the numerical values are taken at the center (X=L/2, Y=0).

	B.C.#1	B.C.#2
Axial Strain	3.039E-05	3.069E-05
Shear Strain	1.138E-07	-1.343E-05
Transverse Strain	-1.752E-05	-1.706E-05

The changes in the strain values for all components are observed in biased cases.
The results from both boundary condition 1 and 2 have similar trend, but the
value is off by about 1 to 4% between the conditions for both axial and transverse
cases, but the most affected strain value due to the boundary condition is the shear
stains. The difference is come from the boundary condition, since the boundary
condition 1 will be restricting the specimen's Y direction displacement at ends
completely while the condition 2 constrains only at the center part of the ends.
Since the strain gage can measure the values within 0.1% of errors, the
differences in the strain values can be detected.

Case 9: Biased Unsymmetric Case
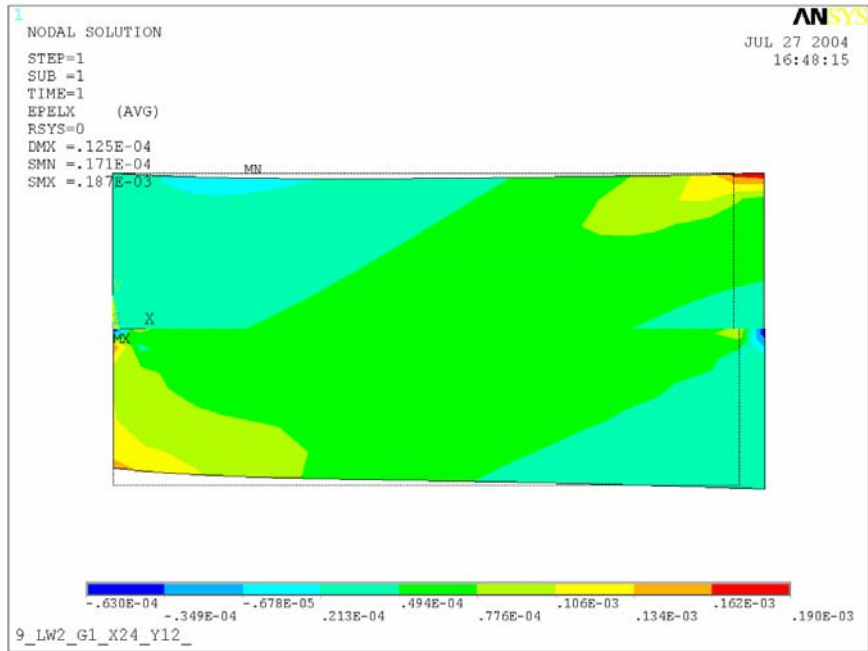
Axial Strain (εX)

Transverse Strain (εY)
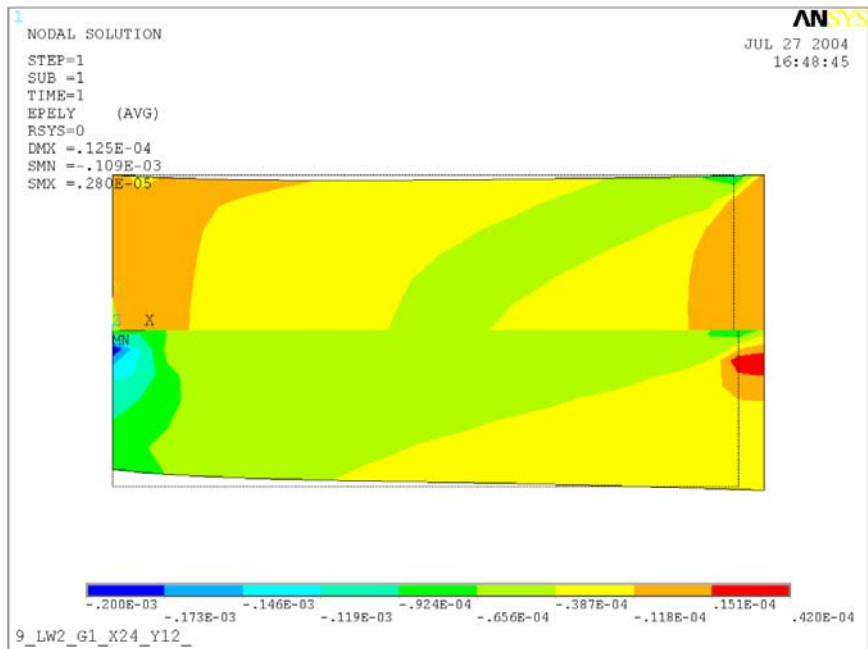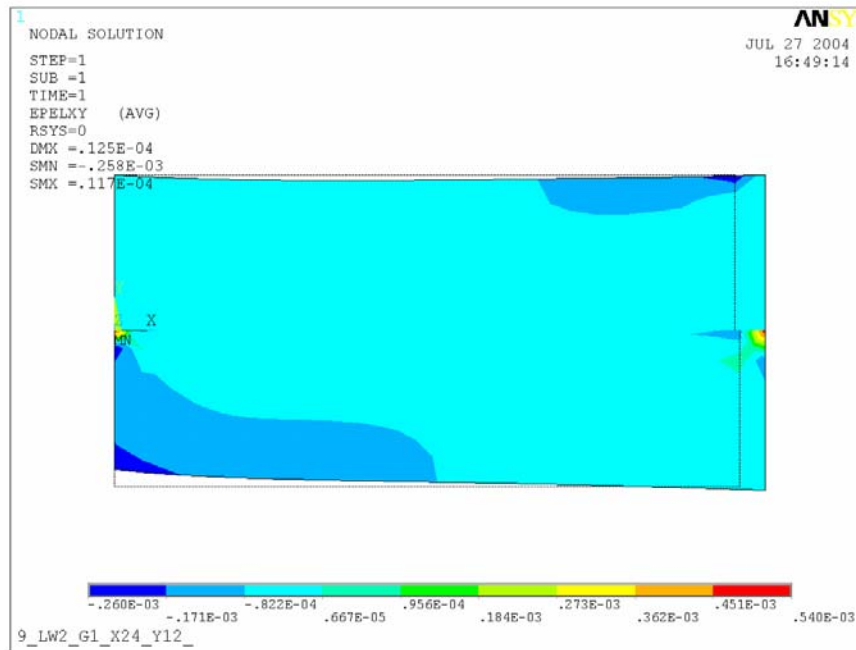
Shear Strain (γXY)

All the numerical values are taken at the center (X=L/2, Y=0).

	B.C.#1	B.C.#2
Axial Strain	6.105E-05	6.501E-05
Shear Strain	-2.011E-05	-6.235E-05
Transverse Strain	-4.031E-05	-4.035E-05

The difference in the values between two different boundary conditions is larger for the case with unsymmetric laminate than the symmetric case. The difference in axial strain is about 5%, but transverse strains values are not far from each other. As observed in the other biased case, #5, the shear strain value has the largest difference between in those two conditions. As noted in the summary of the previous case, the difference in the strain values is caused by the different applied boundary condition and also be notable when the specimen goes under testing.

7 Study 3: Effect of Aspect Ratio

7.1 Overview

This chapter will be focusing on the relationships between the specimen length and the applied boundary conditions for different laminate schedules. From the previous chapter, it is known that the boundary condition setting can affect the results values. The influence of the boundary condition should be diminished as the aspect ratio of the particular specimen increases.
In this section, the aspect ratios are varying from L/W = 2 to 8, and monitor the change in the strain values measured at the center, where the location is X=L/2 and Y=0. However, certain laminate schedules may converge faster than others. This chapter will also identify what types of laminate require the most and least attention.

Like the previous studies, cases 1, 5 and 9 are used to represent the characteristics of the unique cases. The boundary condition #2 is constrained less, meaning that the boundary condition will have smaller effect than the condition 1; therefore, the values for the B.C. #2 should converge faster than the case with B.C. #1 since the constraints set at the ends can cause the differences in the values.
The effect of the boundary condition is diminished if the results from one case match with the other. The components evaluated are axial, transverse, and shear strains at the center of the specimen.

7.2 Results and Summaries

Case 1

Axial Strain (εx)

L/W	B.C. #1	B.C. #2	Plot
2	7.788E-05	7.727E-05	
4	7.736E-05	7.727E-05	
6	7.728E-05	7.727E-05	
8	7.727E-05	7.727E-05	

Shear Strain (γxy)

L/W	B.C. #1	B.C. #2	Plot
2	-0.730E-16	-1.034E-16	
4	1.631E-16	2.189E-16	
6	5.750E-16	5.880E-16	
8	-1.633E-16	-2.972E-16	

Transverse Strain (εy)

L/W	B.C. #1	B.C. #2	Plot
2	-2.284E-05	-2.241E-05	
4	-2.254E-05	-2.241E-05	
6	-2.242E-05	-2.241E-05	
8	-2.241E-05	-2.241E-05	
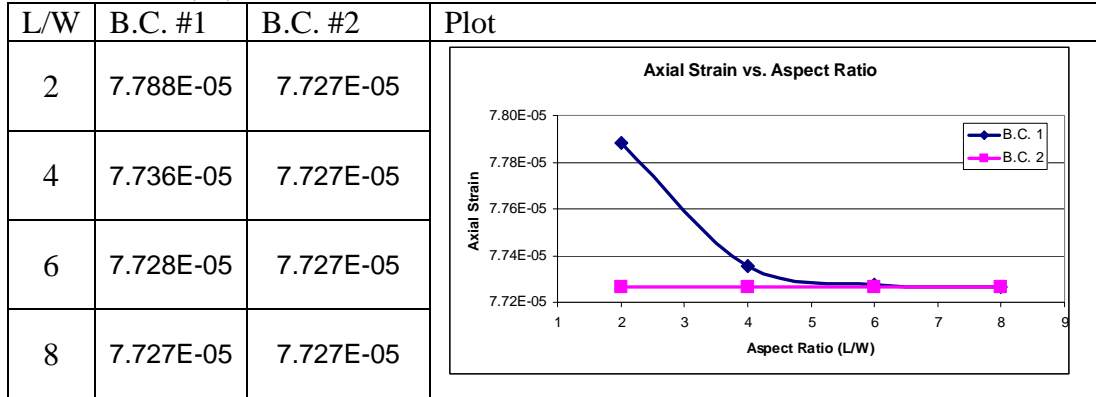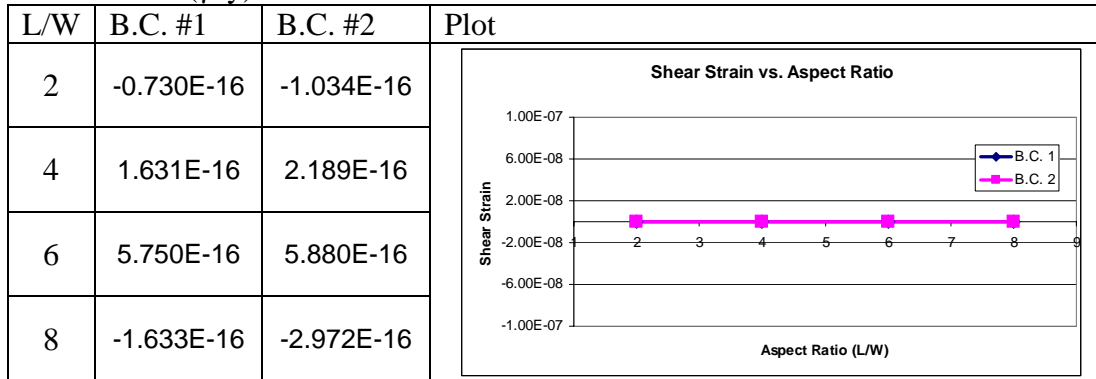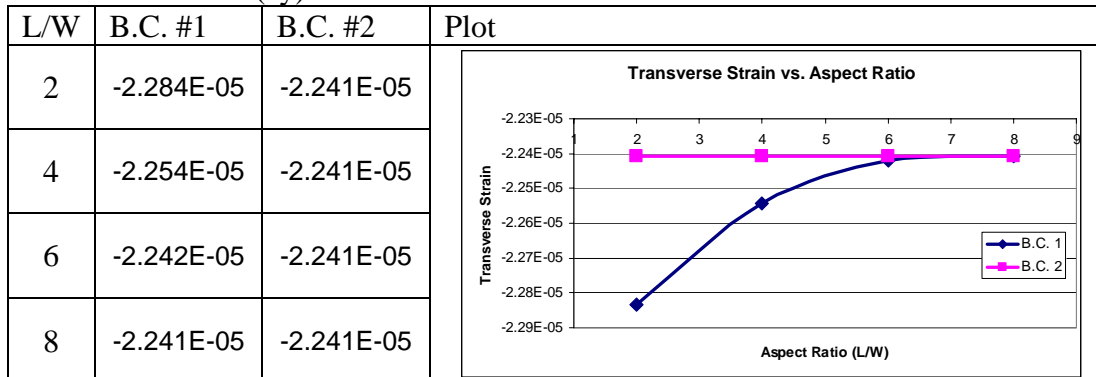
Shear strain is unaffected by the change in aspect ratio since the laminate is symmetric, but axial and transverse strain require the aspect ratio of 6 or higher to get values unaffected by the boundary conditions (0.5% or less at L/W=4).

Case 5

Axial Strain (εx)

L/W	B.C. #1	B.C. #2	Plot
2	3.039E-05	3.069E-05	
4	3.117E-05	3.116E-05	
6	3.137E-05	3.139E-05	
8	3.147E-05	3.149E-05	

Shear Strain (γxy)

L/W	B.C. #1	B.C. #2	Plot
2	0.011E-05	-1.343E-05	
4	-1.231E-05	-1.567E-05	
6	-1.954E-05	-2.064E-05	
8	-2.328E-05	-2.375E-05	

Transverse Strain (εy)

L/W	B.C. #1	B.C. #2	Plot
2	-1.752E-05	-1.706E-05	
4	-1.798E-05	-1.737E-05	
6	-1.722E-05	-1.707E-05	
8	-1.681E-05	-1.675E-05	
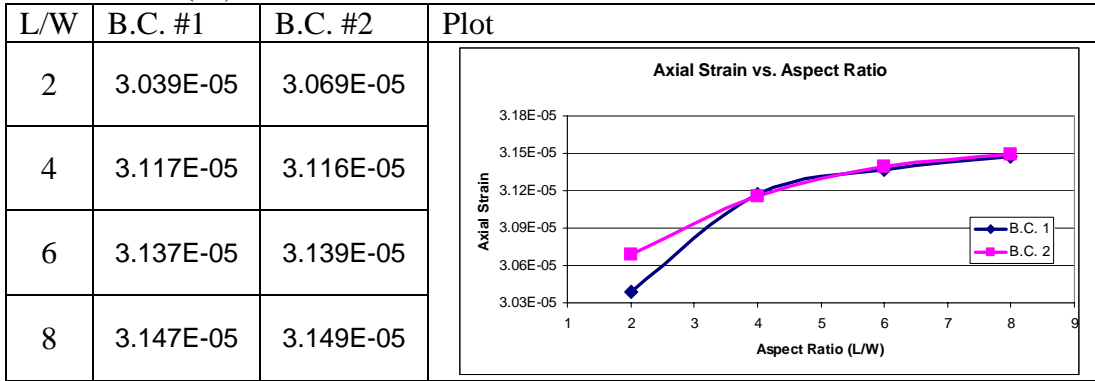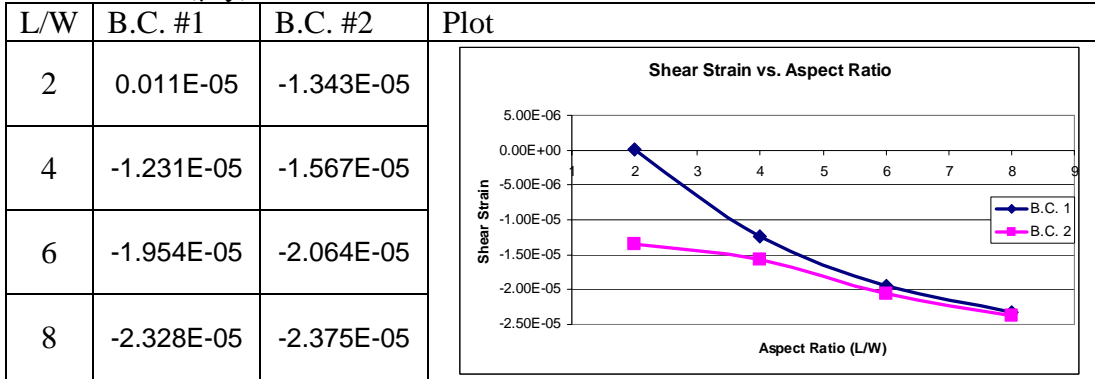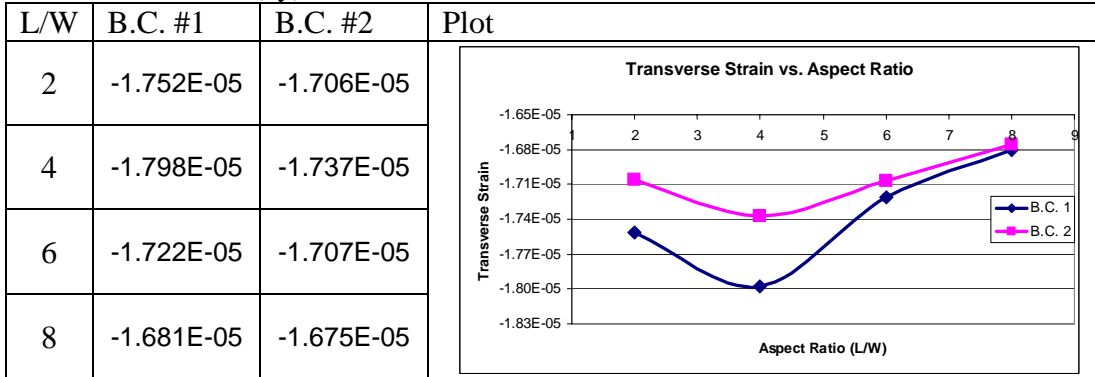
Compared to the symmetric laminate, the boundary conditions play a more important role in the biased case. The cases with a biased laminate change values as the aspect ratio changes no matter what the applied boundary condition; the convergence is also slow in transverse strain. Although the axial and shear strains show the sign of the earlier convergence, the values keep changing; therefore, the case with the biased laminate requires a higher aspect ratio.

Case 9

Axial Strain (εx)

L/W	B.C. #1	B.C. #2	Plot
2	6.105E-05	6.501E-05	
4	7.281E-05	7.352E-05	
6	7.428E-05	7.476E-05	
8	7.542E-05	7.562E-05	

Shear Strain (γxy)

L/W	B.C. #1	B.C. #2	Plot
2	-0.201E-04	-0.624E-04	
4	-0.825E-04	-0.897E-04	
6	-0.991E-04	-1.021E-04	
8	-1.070E-04	-1.084E-04	

Transverse Strain (εy)

L/W	B.C. #1	B.C. #2	Plot
2	-4.031E-05	-4.035E-05	
4	-4.893E-05	-4.633E-05	
6	-4.411E-05	-4.372E-05	
8	-4.276E-05	-4.255E-05	
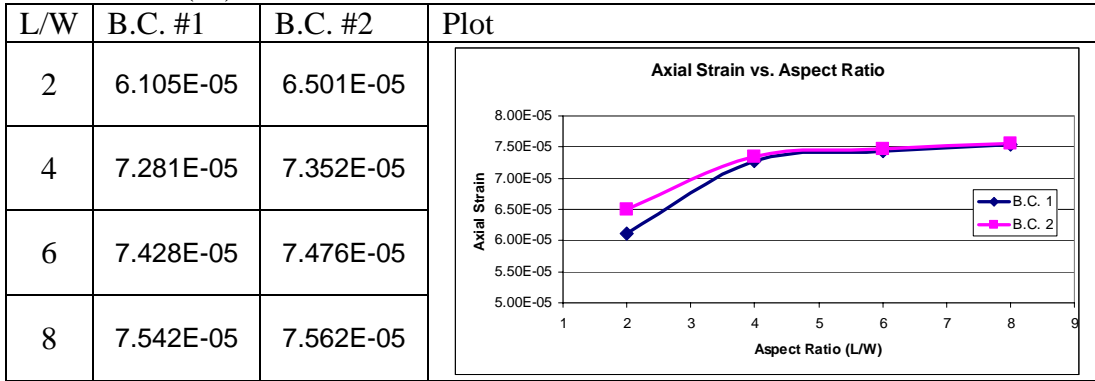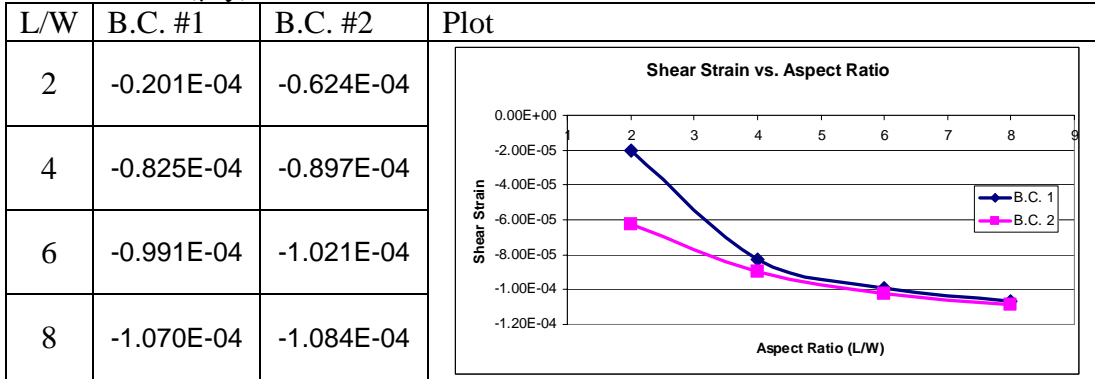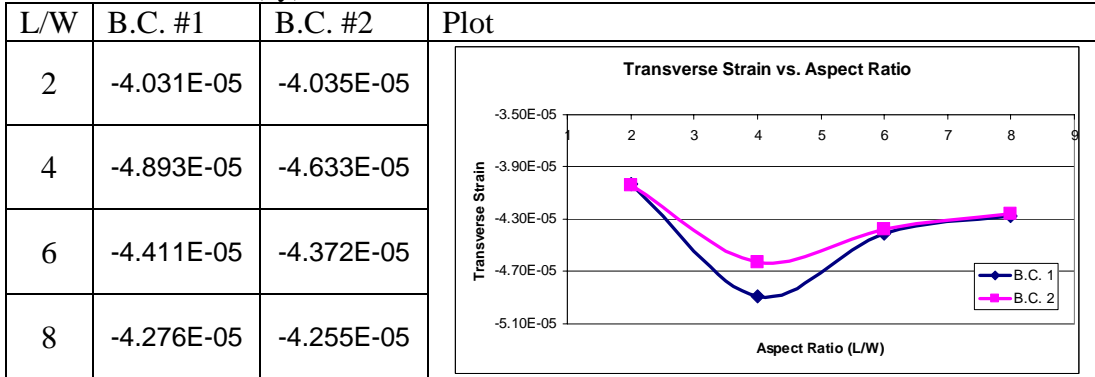
Compared to the biased case, this case has a faster convergence rate. Both axial and transverse strains have a matching result for L/W=4 or higher, the shear stain still requires a ratio around 6.

Summaries

Case 1

Axial and Transverse cases show convergence while the shear strain is constant The values for both strain values will not be affected by the boundary conditions after the L/W ratio is 6 or higher.

Case 5

All three strains converge as the aspect ratios change. The Axial strain will be converged at first around L/W=4; the transverse strain value remains non-zero due to the biased layup.

Case 9

Shear strain is the only component which has difference in the values from two different boundary conditions. The other components are virtually unaffected. However, the result values changes as the aspect ratio changes. The transverse strain value remains non-zero due to the biased layup.

Unlike the balanced case, the biased case will have shifts in the strain values during the changes in aspect ratios. The balanced case shows that one of the solutions is close to the other while the biased cases follow a similar trend but the both of the curves will be converging toward a certain value.

8 Conclusions

Over this study, six basic cases and three additional cases were tested in ANSYS to investigate the capabilities of the FEA program, as well as the behaviors of the composite laminates in different laminate schedules, boundary conditions, and the aspect ratios.

During the basic study, ANSYS has proven that the program can handle single layer unaxial, multi layer specially orthotropic, single layer biased, and multi layer biased cases. In all cases, axial, transverse, and shear strain were compared in different boundary conditions and also in different aspect ratios. From the result, strain values from the all four cases are following the CLT results, indicating that the ANSYS can provide reliable results in various cases. The distribution of the strain values were observed from the ANSYS contour plots, and they indicate that the both single layer uniaxial and multi layer specially orthotropic case can be tested at any aspect ratio, while the case with the biased laminates require the minimum aspect ratio of 6 to have the strain values unaffected by the boundary condition.

The additional basic cases were the case with unsymmetric laminate, which causes vertical displacements and also the case with coupling, which will have a twist in the laminate. When the strain values were compared with the value determined from classical lamination theory method, they are close matches to each others. However, the strain values determined in the boundary condition with both ends constrained have different result from the condition with one free end.

In main part of the project, three unique cases were chosen, and studied from the view point of laminate schedules, boundary conditions, and aspect ratios.

By studying the laminate schedules and their strains, the first case has balanced symmetric, second case is biased symmetric and the last case has the biased, unsymmetric laminate schedules. The case with balanced laminate has uniform strain values while the case with biased laminate shows change in strain values along the length due to the offset angle of the fiber. The change in the strain values are largest at for the case with biased, unsymmetric laminate.

After identifying the basic characteristics of the laminate, each set was compared in two different boundary conditions to observe the trend in strain values.
In general, the axial and transverse strain values are affected most for the case with uniaxial layup, while biased cases have large difference in the axial and shear strain values between the boundary condition 1 and 2. The difference caused in shear strain is believed to be relationships between the offset fiber direction and the applied boundary condition. This means that the cases with biased laminate schedules are

affected by the boundary condition more than the cases with balanced laminate schedule.

Once those items were studied, the aspect ratios were changed for the chosen cases to determine the influence of the boundary conditions for the aspect ratio. From the findings in the tests, the aspect ratio of six or higher will have the strain values form different boundary condition to match.

In conclusion, the minimum aspect ratio of six should have the adequate condition for measuring the strain values for any laminate condition, as well as the strain values under tensile load.

Appendix A: Classical Lamination Theory

Classical Lamination Theory (CLT)

The Classical Lamination Theory (CLT) is a simplified method to approximate the mechanical behavior of certain types of composite laminates. Since this is an approximation, several assumptions must be taken into account. Reference #3 provides a detail description of CLT.

Q and Q bar matrix

Unlike an isotropic stiffness matrix used for conventional materials, composite materials that are orthotropic materials have directional stiffness properties.

Q Matrix $[Q]$

The Q matrix is the basis for creating the stiffness matrix for orthotropic laminates.

$$[Q] = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{21} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix}$$

Each component for the Q matrix can be derived as shown below.

$$Q_{11} = \frac{E_1}{1 - \upsilon_{12}\upsilon_{21}}, \; Q_{22} = \frac{E_2}{1 - \upsilon_{12}\upsilon_{21}}, \; Q_{12} = \upsilon_{21}Q_{11} = \upsilon_{12}Q_{22}, \; Q_{66} = G_{12} \text{ and}$$

v12 is transverse Poisson's ratio and can be derived as, $\upsilon_{21} = \upsilon_{12}\dfrac{E_{22}}{E_{11}}$

The Q matrix relates stress to strain in the local coordinate system.
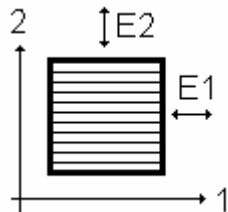
Local coordinate system

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & Q_{16} \\ Q_{21} & Q_{22} & Q_{26} \\ Q_{61} & Q_{62} & Q_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix}$$

Q bar Matrix $\left[\overline{Q}\right]$

The Q bar matrix is a stiffness matrix that is transformed into the global coordinate system (X, Y) from the local coordinate system (1, 2).

$$\overline{Q}_{11} = Q_{11}\cos^4\theta + 2(Q_{12} + 2Q_{66})\sin^2\theta\cos^2\theta + Q_{22}\sin^4\theta$$

$$\overline{Q}_{22} = Q_{11}\sin^4\theta + 2(Q_{12} + 2Q_{66})\sin^2\theta\cos^2\theta + Q_{22}\cos^4\theta$$

$$\overline{Q}_{12} = (Q_{11} + Q_{22} - 4Q_{66})\sin^2\theta\cos^2 + Q_{12}\left(\sin^4\theta + \cos^4\theta\right)$$

$$\overline{Q}_{66} = (Q_{11} + Q_{22} - 2Q_{12} - 2Q_{66})\sin^2\theta\cos^2\theta + Q_{66}\left(\sin^4\theta + \cos^4\theta\right)$$

$$\overline{Q}_{16} = (Q_{11} + Q_{22} - 4Q_{66})\sin\theta\cos^3\theta + (Q_{12} - Q_{22} + 2Q_{66})\sin^3\theta\cos\theta$$

$$\overline{Q}_{26} = (Q_{11} + Q_{22} - 4Q_{66})\sin^3\theta\cos\theta + (Q_{12} - Q_{22} + 2Q_{66})\sin\theta\cos^3\theta$$
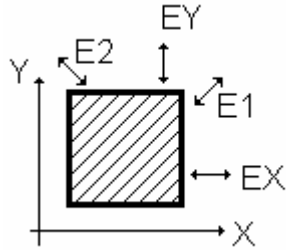
Relationships of the global coordinate system and local coordinate system

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{16} \\ \overline{Q}_{21} & \overline{Q}_{22} & \overline{Q}_{26} \\ \overline{Q}_{61} & \overline{Q}_{62} & \overline{Q}_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

Stiffness Matrix (A, B and D Matrices)

After determining the Q bar matrix, the relationship between the loadings and resultant strains and curvatures can be determined by using the stiffness matrix.

This matrix consists of three different matrices; A, B and D, and the relationship is shown below.

$$\begin{bmatrix} N \\ M \end{bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{bmatrix} \varepsilon^0 \\ \kappa \end{bmatrix}$$

A matrix (Extensional Stiffness)

The A matrix is described as $[A] = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{21} & A_{22} & A_{26} \\ A_{61} & A_{62} & A_{66} \end{bmatrix}$ and each component of the

A matrix is defined as, $A_{ij} = \sum_{k=1}^{N} \left(\overline{Q}_{ij} \right)_k \left(z_k - z_{k-1} \right)$ with unit of (N/m).

By using $[\overline{Q}]$, the A matrix can be determined in the global coordinate system.

Z is the distance measured from the center of the laminate and also a component of thickness. N is the number of layers.
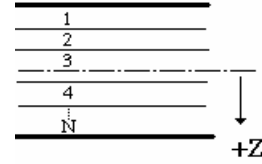
B matrix (Coupling Stiffness)

The B matrix is similar to the B matrix. The B matrix is use to determine the effect on displacement due to moment, and effect of bending due to the plane loadings. The unit for this matrix is (N)

$[B] = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{21} & B_{22} & B_{26} \\ B_{61} & B_{62} & B_{66} \end{bmatrix}$ and each component can be described as,

$B_{ij} = \frac{1}{2} \sum_{k=1}^{n} \left(\overline{Q}_{ij} \right)_k \left(z_k^2 - z_{k-1}^2 \right)$

D matrix (Bending Stiffness)
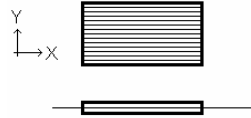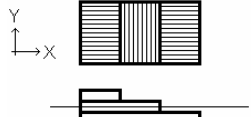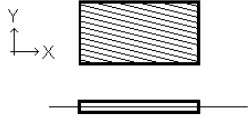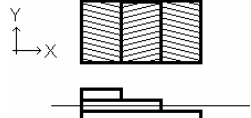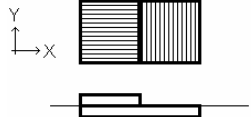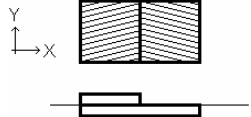
The D matrix uses the same $[\overline{Q}]$, but the z values are at the power of 3 and the multiplier is 1/3. The D matrix is used to describe the relationship between the Moments applied and the curvature of the coupon. The unit is (N-m)

$[D] = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{21} & D_{22} & D_{26} \\ D_{61} & D_{62} & D_{66} \end{bmatrix}$ and $D_{ij} = \frac{1}{3} \sum_{k=1}^{n} \left(\overline{Q}_{ij} \right)_k \left(z_k^3 - z_{k-1}^3 \right)$

III

Appendix B: Basic Studies

Basic Cases to Be Studied

Four basic cases were chosen to validate ANSYS capabilities for analyzing composite material. Those cases can easily be compared with the values acquired from Classical Lamination Theory. Each case was evaluated using aspect ratios of 2, 4, 6 and 8 respectively.

Case#	Configuration	Laminate Description
1	Single layer uniaxial orthotropic	0° carbon 0.000125m
2	Multilayer Specially orthotropic	0° carbon 0.000125m 90° carbon 0.000125m 0° carbon 0.000125m
3	Single layer biased	-20° carbon 0.000125m
4	Multi layer biased	-20° carbon 0.000125m +20° carbon 0.000125m -20° carbon 0.000125m
5	Balanced Unsymmetric	0° carbon 0.000125m 90° carbon 0.000125m
6	Balanced with Coupling	+20° carbon 0.000125m -20° carbon 0.000125m

Boundary Conditions and Loading

There are a total of four boundary conditions defined to perform the basic studies. The boundary conditions 1 and 2 are the same conditions as used in the main part of analysis in this paper, but the conditions 3 and 4 are specifically used to verify the laminated material capabilities in ANSYS (Used fro case 5 and 6).

Boundary Condition #1
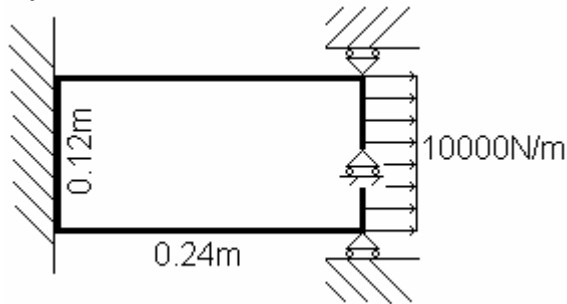
This boundary condition allows no movement for any directions at X = 0m. At X = 0.24m, the constraints are set in similar ways, but displacement in the X direction is allowed. There will be no Z direction displacement at either end.

Boundary Condition #2
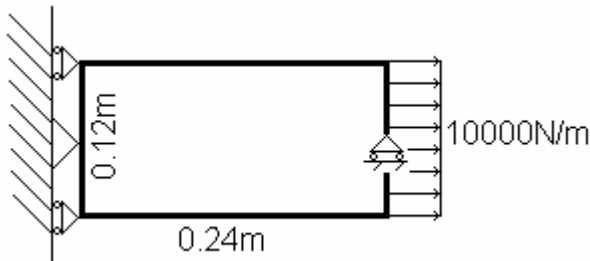
The other boundary condition still does not allow movement in the Y direction at both ends but is limited to the center (Y = 0). There will be no Z direction displacements at either end.

Boundary Condition #3 (for Curvature study)

This boundary condition allows the coupon to bend vertically due to the curvature generated from the laminate under load.

The figure shows the Z direction Displacement.
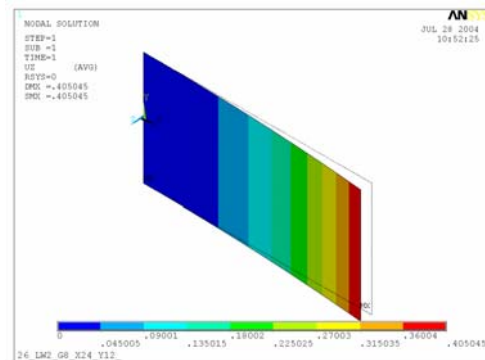
Boundary Condition #4 (for coupling Study)

The last condition is to allow twist in the coupon. This boundary condition is similar to the boundary condition #2, but the displacement in the Z direction is allowed.

The figure shows the Z direction Displacement.
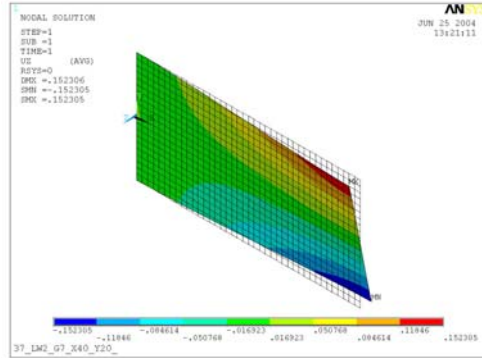
Notes:

In all cases, one of the ends at X = 0.24m is set so that the entire end moves the same amount in the X direction (UX is constant at X=0.24).

Only one element size of 0.01m by 0.01m is used for both case since it can provide enough elements in both the x and y directions.
Each model is being tested for both boundary conditions.
There is one loading set for the study. The Nxx = 10,000N/m is used in CLT while 1,200N of the force is applied for the ANSYS models to create the same loading condition.

Utilizing the Classical Lamination Theory (CLT)

A Matrix for Cases

After determining the laminate, layup order and thickness of each ply, and the material properties used in the set, the A matrix has been determined.

Case 1: Uniaxial, single layer

$$[A] = \begin{bmatrix} 1.509E+7 & 3.286E+5 & 0 \\ 3.286E+5 & 1.173E+6 & 0 \\ 0 & 0 & 5.838E+5 \end{bmatrix} \text{(N/m)}$$

Case 2: Uniaxial Multi layer

$$[A] = \begin{bmatrix} 3.135E+7 & 9.858E+5 & 0 \\ 9.858E+5 & 1.744E+7 & 0 \\ 0 & 0 & 1.751E+6 \end{bmatrix} \text{(N/m)}$$

Case 3: Biased, single layer

$$[A] = \begin{bmatrix} 1.209E+7 & 1.700E+6 & -3.871E+6 \\ 1.700E+6 & 1.431E+6 & -6.027E+5 \\ -3.871E+6 & -6.027E+5 & 1.955E+6 \end{bmatrix} \text{(N/m)}$$

VI

Case 4: Biased, multi layer

$$[A] = \begin{bmatrix} 3.627\text{E}+7 & 5.100\text{E}+6 & -3.871\text{E}+6 \\ 5.100\text{E}+6 & 4.293\text{E}+6 & -6.027\text{E}+5 \\ -3.871\text{E}+6 & -6.027\text{E}+5 & 5.865\text{E}+6 \end{bmatrix} \text{ (N/m)}$$

Case 5: Balanced, unsymmetric

$$[A] = \begin{bmatrix} 1.627E+7 & 6.571E+5 & 0 \\ 6.571E+5 & 1.627E+7 & 0 \\ 0 & 0 & 1.168E+6 \end{bmatrix} \text{ (N/m)}$$

Case 6: Balanced, crossply

$$[A] = \begin{bmatrix} 2.419\text{E}+7 & 3.399\text{E}+6 & 0 \\ 3.399\text{E}+6 & 2.861\text{E}+6 & 0 \\ 0 & 0 & 3.910\text{E}+6 \end{bmatrix} \text{ (N/m)}$$

Strain Values from CLT

The only load applied is a force in X direction with magnitude of 1,200N. There is no moment applied. Since the width of the coupons is 0.12m, the Nxx component can be set as 10,000N/m. The other components (Nyy and Nxy) are zero.

The Load matrix is defined as, $[N] = \begin{bmatrix} 10000 \\ 0 \\ 0 \end{bmatrix}$ (N/m)

From the equation $[N] = [A]\{\varepsilon\}$, the strain values are determined from
$[\varepsilon] = [\text{A}]^{-1} * [N]$

By using the formula, the results for each case are

Case 1:

$$[\varepsilon] = \begin{bmatrix} 6.668*10^{-4} \\ -1.868*10^{-4} \\ 0 \end{bmatrix} \text{ (m/m)}$$

Case 4:

$$[\varepsilon] = \begin{bmatrix} 3.521*10^{-4} \\ -3.900*10^{-4} \\ 1.972*10^{-4} \end{bmatrix} \text{ (m/m)}$$

Case 2:

$$[\varepsilon] = \begin{bmatrix} 3.195*10^{-4} \\ -1.806*10^{-5} \\ 0 \end{bmatrix} \text{ (m/m)}$$

Case 5:

$$[\varepsilon] = \begin{bmatrix} 6.156*10^{-4} \\ -2.486*10^{-5} \\ 0 \end{bmatrix} \text{ (m/m)}$$

Case 3:

$$[\varepsilon] = \begin{bmatrix} 2.578*10^{-3} \\ -9.412*10^{-4} \\ 4.933*10^{-3} \end{bmatrix} \text{ (m/m)}$$

Case 6:

$$[\varepsilon] = \begin{bmatrix} 4.962*10^{-4} \\ -5.895*10^{-4} \\ 0 \end{bmatrix} \text{ (m/m)}$$

Equivalent Moduli

After laminate is stacked, the overall mechanical properties depend on the schedule. By using the CLT A-matrix, the basic values such as Ex, Ey, Gxy and vxy can be calculated. However, if the coupon is coupled or unbalanced, this method cannot be used. (This method cannot be used if D_{16} and D_{26} are non-zero value, so this cannot be used for case 3 and case 4)

$$Ex = \frac{A_{11}A_{22} - A_{12}^2}{tA_{22}}, \ Ey = \frac{A_{11}A_{22} - A_{12}^2}{tA_{11}}, \ Gxy = \frac{A_{66}}{t} \text{ and } vxy = \frac{A_{12}}{A_{22}}$$

Case	Ex (Pa)	Ey (Pa)	Gxy (Pa)
1	$1.200*10^{11}$	$9.330*10^9$	$4.670*10^9$
2	$8.347*10^{10}$	$4.642*10^{10}$	$4.670*10^9$
3	$3.378*10^{10}$	$9.495*10^9$	$5.704*10^9$
4	$7.600*10^{10}$	$9.531*10^9$	$1.453*10^{10}$
5	$2.922*10^{10}$	$2.922*10^{10}$	$4.670*10^9$
6	$4.675*10^{10}$	$9.513*10^9$	$8.188*10^9$

Element Size Selection

The adequate size and number of the elements can be determined by trial and error by comparing the results as a function of the number of elements.

Smaller element size provides smoother curvature, more flexibility and also gives a wider valid range of values in case the model has stress concentrations.

Example of the effects of the element size
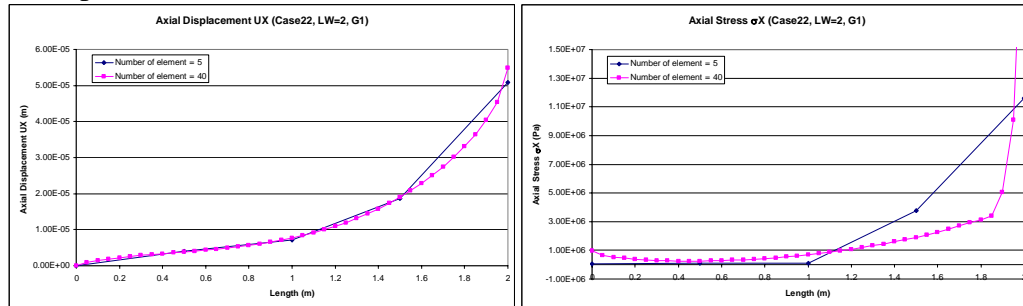
As seen on the left chart, using fewer numbers of elements will provide less detail, as well as making the coupon stiffer, resulting in a lower displacement value than the case with more elements. In addition, if there is a case which has divergence near the ends, the model with a larger element size will start diverging at earlier stage while the finer meshed coupon will have a wider valid element range. This effect is shown in the right hand side plot.

Result from Case 1 and 2: Balanced, Symmetric Cases

Case 1: Single Layer Uniaxial Case

Plots of Axial Strain (εx) measured along the length

Numerical results from ANSYS are plotted here. The values are taken at three places along the length at Y=W/4, Y=0 and Y=-W/4.
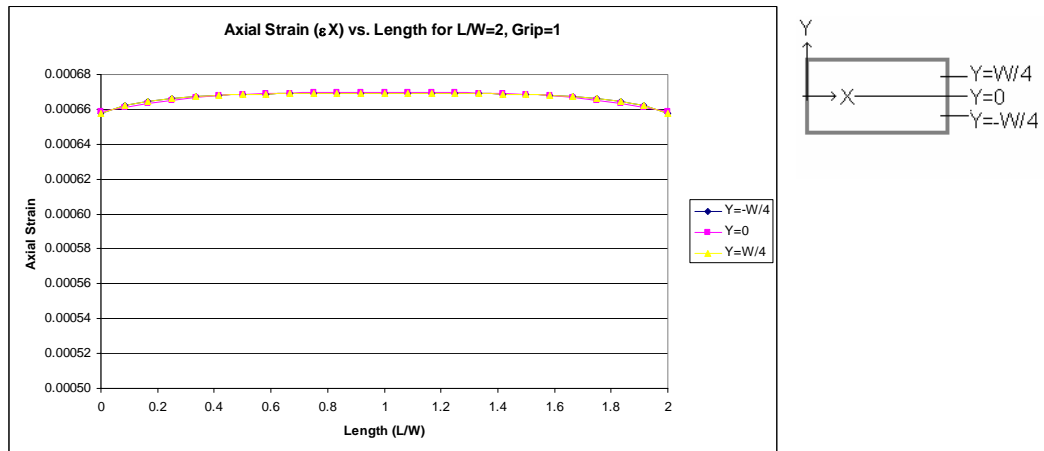
Axial Strain plot for B.C. 1 condition. All three curves have the same value at all lengths, but the value decreases near the ends due to the boundary condition.
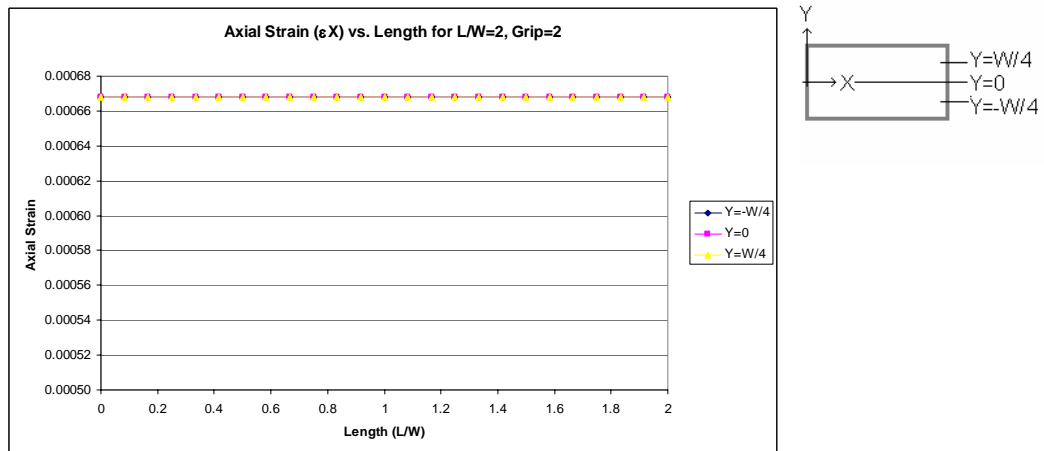
Axial Strain plot for B.C. 2 condition, the values are uniform for all three lines along the entire range.

Axial Strain (εx)

Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	0.6697E-03	0.6678E-03	
4	0.6685E-03	0.6678E-03	
6	0.6680E-03	0.6678E-03	
8	0.6678E-03	0.6678E-03	
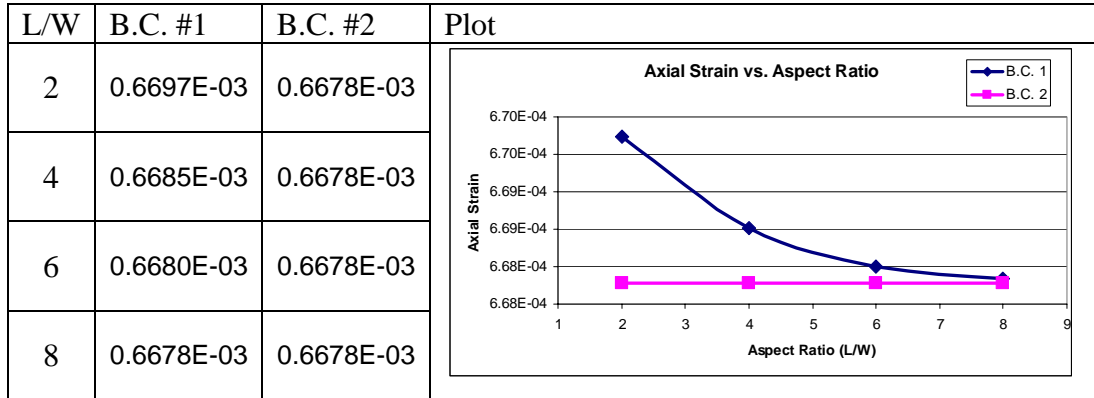
Numerical values are taken at the central portion of the specimen. This is the tabulated result taken from ANSYS at $(X, Y) = (L/2, 0)$. While BC2 has a constant value at any aspect ratio, the values for BC1 decreases as the L/W ratio get large. The values for both cases converges at L/W=8, meaning that the effect of boundary conditions diminishes. The value from the CLT method is $0.6668*10^{-3}$ so the both cases have the values matching with this method.

Transverse Strain (εY)

Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	-1.863E-04	-1.870E-04	
4	-1.881E-04	-1.870E-04	
6	-1.873E-04	-1.870E-04	
8	-1.871E-04	-1.870E-04	
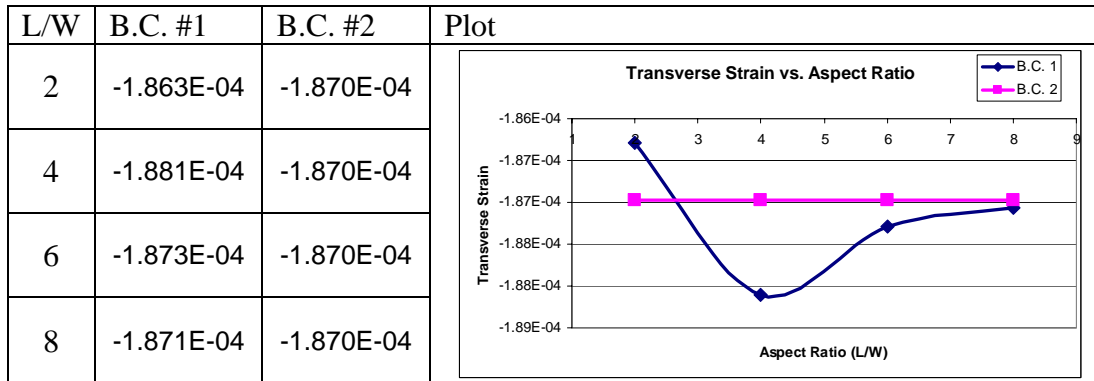
By using the CLT method, the transverse stain value for this case was $-0.1868*10^{-3}$ in this case both results are matching with the calculated solution.

<u>Shear Strain (γXY)</u>

Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	-7.483E-16	-1.098E-15	
4	1.664E-15	2.590E-15	
6	2.456E-15	2.240E-15	
8	-1.396E-14	-1.685E-14	
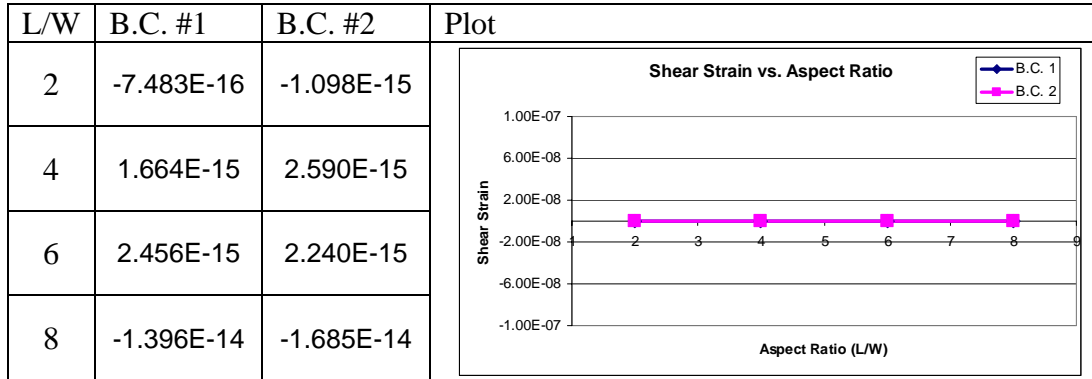
The CLT method shows the value of 0 and the result from both boundary
condition is the same result and valid for all aspect ratios

Case 2: Multi-layer Specially Orthotropic Case

Axial Strains (εx)
    Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	0.3201E-05	0.3200E-05	
4	0.3200E-05	0.3200E-05	
6	0.3200E-05	0.3200E-05	
8	0.3200E-05	0.3200E-05	
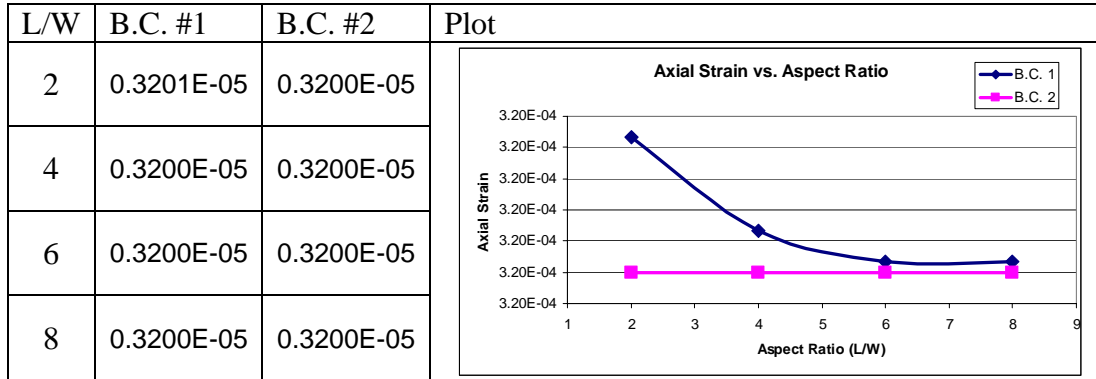
The values were taken from the central portion of the coupon (X, Y) = (L/2, 0). The values at any aspect ratios for B.C. #2 are the same, while B.C. #1 values are still slow to converge for changes in the length of the specimen. The values for both cases match for L/W=6 or higher. Compared with case 1, the values converge for a lower aspect ratio. The result from the CLT method shows $0.3195*10^{-5}$, which is closely matching with the resultant values in this case. Relationship between aspect ratio and Grip condition is significant. Note the rapid convergence for the B.C. #1 while the B.C. #2 has straight line.

Transverse Strains: (εY)
    Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	-1.816E-05	-1.811E-05	
4	-1.812E-05	-1.811E-05	
6	-1.811E-05	-1.811E-05	
8	-1.811E-05	-1.811E-05	
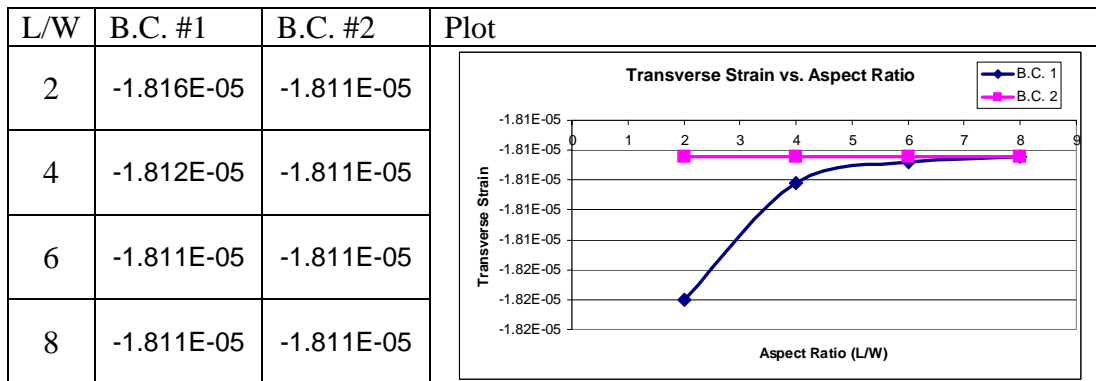
The value calculated using CLT method is $-0.1806*10^{-6}$, since the case is symmetric so the layup is balanced. Therefore, the strain value was very much constant.

Shear Strains (γXY)

Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	-4.548E-16	-5.157E-16	
4	1.316E-16	3.091E-16	
6	2.017E-15	2.139E-15	
8	-7.715E-15	-8.052E-15	
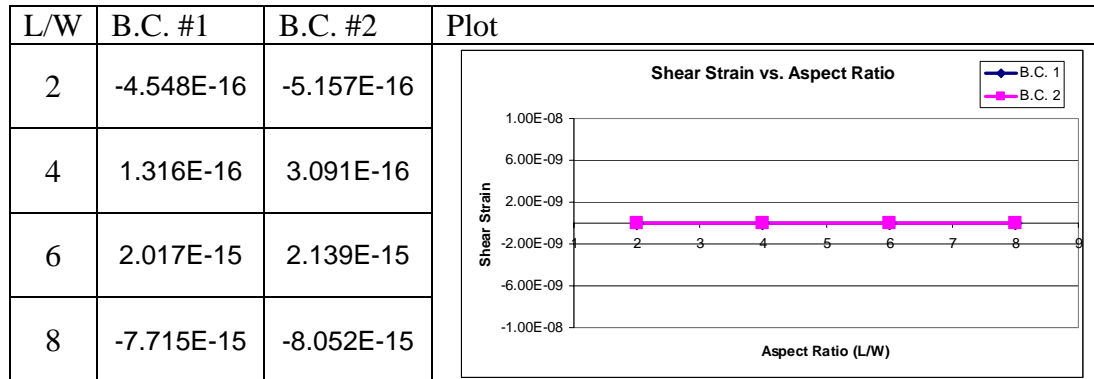
The boundary condition does not affect the result since the laminate is symmetric and the values acquired are zero, which is the same as the result value based on the CLT method.

Result from Case 3 and 4: Biased Cases

Biased Case Study

Unlike the uniaxial case, the biased cases behave differently, and several studies have been done to determine the stress and strain states of the coupons. Based on reference #1, the validity of ANSYS results were checked.

A total of four plots are given in the reference #1, the shear stress (τxy) and strain (γxy) for L/W = 2 and L/W = 4 with rigid ends (Boundary Condition 1) were tested.
Although the dimensions are unknown and there is no specific loading condition is described expect a "unit load", the values should provide a reference for the values from ANSYS. This means objective was to see if the ANSYS model produces similar qualitative results.
The values are measured at 45 places (Every L/8 in the X direction and W/4 in the Y direction), and tabulated so that the data can be used to compare the values with ANSYS result.
After the values were taken from the plots, two ANSYS case were conducted. The condition for the two cases were L/W=2 and L/W=4 for BC 1.
The results were compared with the values from the plots. In general, the values from the plots are multiplied by a certain constant and if necessary, the values are adjusted by a constant.

Shear Stress for L/W=2

(b) Iso-shear stress $\mathcal{T}_{xy}$ x 10^{-3}
Non-rotating
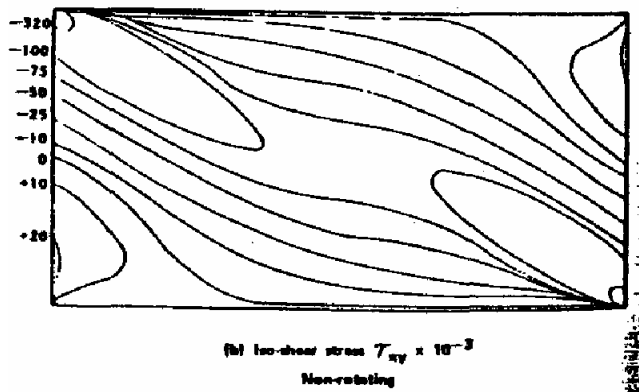
X	Y=W/2	Y=W/4	Y=0
0.00	10	20	0
0.25	5	5	-25
0.50	5	-10	-75
0.75	0	-25	-90
1.00	0	-50	-80
1.25	0	-55	-90
1.50	0	-100	-75
1.75	0	-150	-25
2.00	-320	-60	0

Shear Stress Approximate Values and Plot from Reference 1 (L/W=2)

Shear Stress (ANSYS vs. Readout) L/W=2

Shear Stress

Length

Legend:
- Y=-W/2 (ANSYS)
- Y=-W/4 (ANSYS)
- Y=0 (ANSYS)
- Y=W/2 (Ref. 1)
- Y=W/4 (Ref. 1)
- Y=0 (Ref. 1)
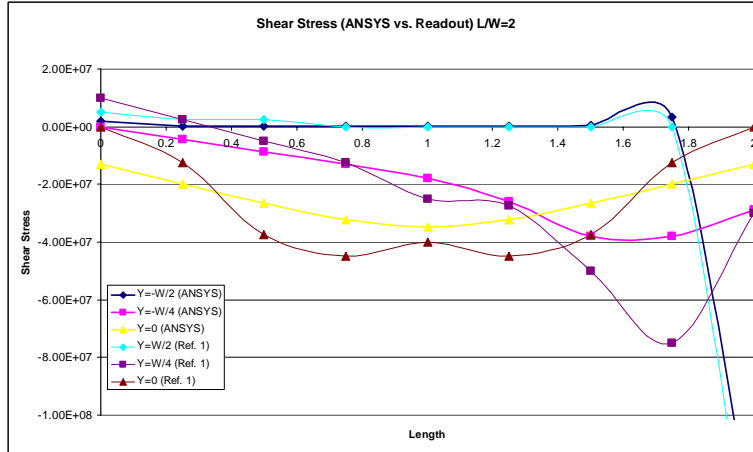
Values for Y=W/2 and 0 are multiplied by $5.0*10^5$. The values for Y=W/4 were multiplied by $-5.0*10^5$. Values from ANSYS follow the same trend with the values from reference #1.

Shear Stress for L/W=4
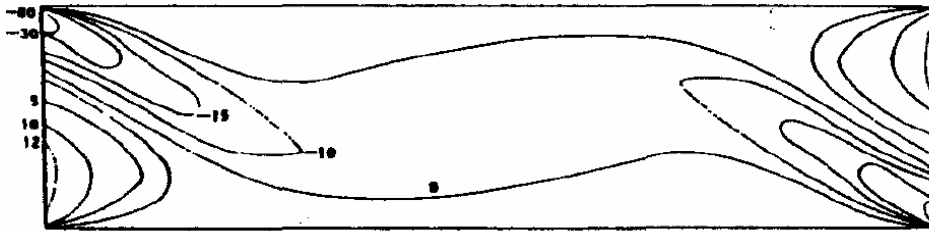
(b) In-shear stress $T_{xy} \times 10^{-4}$

Shear Stress Values Plot for L/W=4

From the contour plot above, these are the value determined

X	Y=W/2	Y=W/4	Y=0
0.0	-80	-10	10
0.5	0	-14	-10
1.0	0	0	0
1.5	0	0	0
2.0	0	-2	-5
2.5	0	-5	0
3.0	0	-5	0
3.5	1	3	-10
4.0	10	15	10
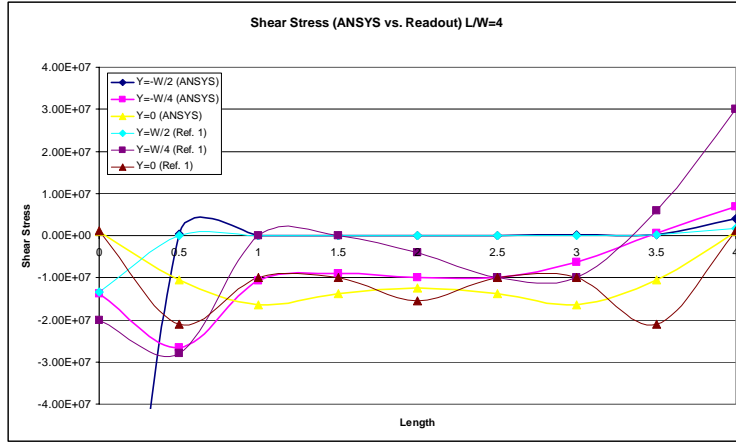
XVI

Shear Stress (ANSYS vs. Readout) L/W=4

Similar to the case for L/W=2, the values Y=W/2 are closely matching with the values read from the plot. The curves for Y=0 are indicating a similar trend.

Shear Strain for L/W=2
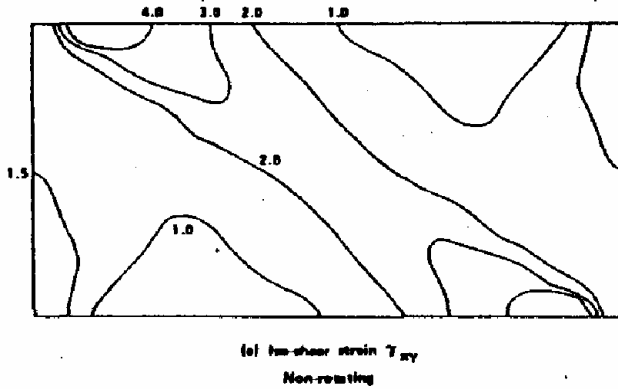
X	Y=W/2	Y=W/4	Y=0
0.00	1.3	1.3	1.5
0.25	0.8	1.2	1.7
0.50	0.0	0.8	1.7
0.75	0.5	2.5	2.0
1.00	1.0	1.8	2.0
1.25	2.0	2.5	2.0
1.50	3.5	2.8	1.7
1.75	4.5	1.8	1.7
2.00	1.8	1.7	1.5

Shear Strain Approximate Values and Plot from Reference 1 (L/W=2)

Shear Strain (ANSYS vs. Readout) L/W=2
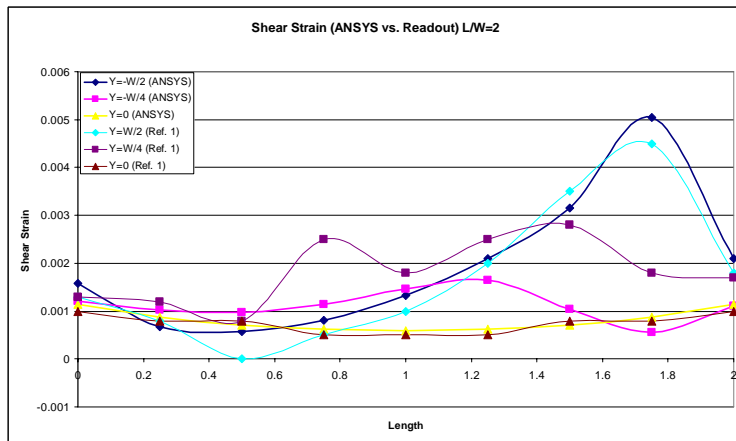
Similar to the case for L/W=2, the values for Y=W/2 are closely matching with the values read from the plot. The curves for Y=0 are indicating a similar trend.

XVII

Shear Strain Plot for L/W=4.

(a) % iso-shear strain γ_{xy}
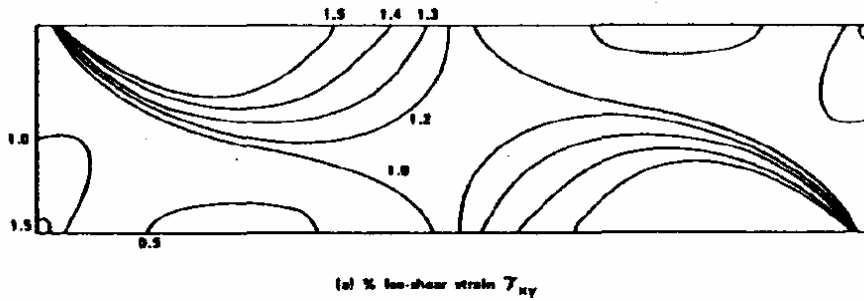
Shear Strain Values for L/W=4 approximated from the isobar plot above

X	Y=W/2	Y=W/4	Y=0
0.0	1.0	1.0	1.0
0.5	1.5	1.5	1.1
1.0	1.5	1.5	1.3
1.5	1.4	1.4	1.2
2.0	1.2	1.2	1.2
2.5	0.7	1.0	1.2
3.0	0.0	0.8	1.3
3.5	0.5	0.8	1.1
4.0	1.5	1.2	1.0
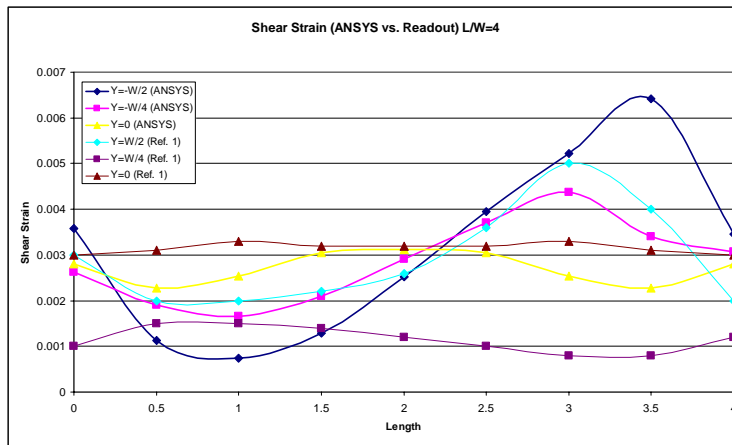
Similar to the case for L/W=2, the values Y=W/2 are closely matching with the values readout from the plot. The curves for Y=0 are indicating a similar trend.

From these findings the values from ANSYS can be said to be reasonable.

Case 3: Single Layer Biased Case

Plots of Axial Strains (εx) measured along the length

Strain values are measured for both boundary condition 1 (Grip1) and 2 (Grip2).

Axial Strain (εX) vs. Length for Case3, L/W=2, Grip1
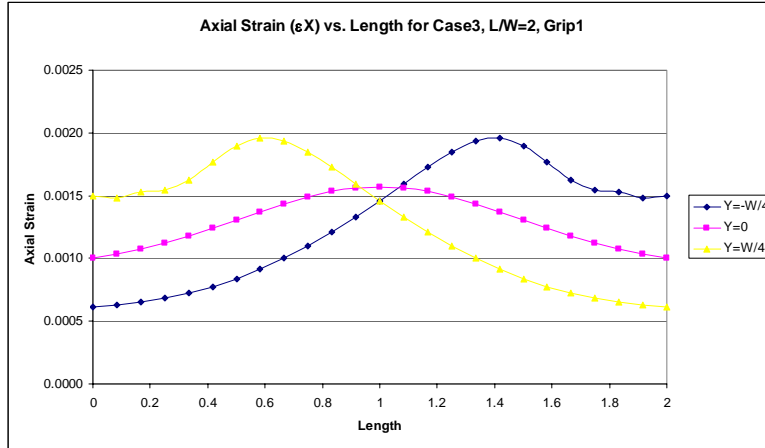
Axial Strain plot for Y=W/4, 0 and –W/4 shown. The curves for W/4 and –W/4 are a mirror image at the center, while data taken at Y=0 is symmetric.

Axial Strain (εX) vs. Length for Case3, L/W=2, Grip2
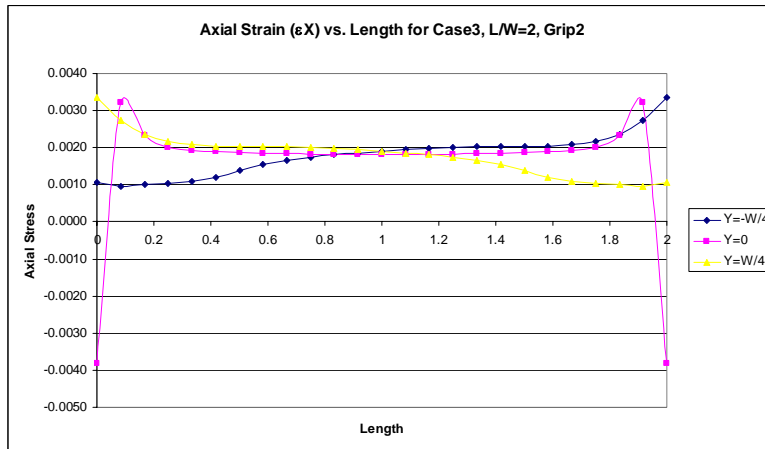
The larger divergences for Y=0 at both end is observed. This is due to the constraints at the center.

Axial Strain values measured at specimen center (X=L/2, Y=0)

L/W	B.C. #1	B.C. #2	Plot
2	$1.5723*10^{-3}$	$1.8207*10^{-3}$	
4	$2.1928*10^{-3}$	$2.2216*10^{-3}$	
6	$2.2479*10^{-3}$	$2.2692*10^{-3}$	
8	$2.3002*10^{-3}$	$2.3084*10^{-3}$	
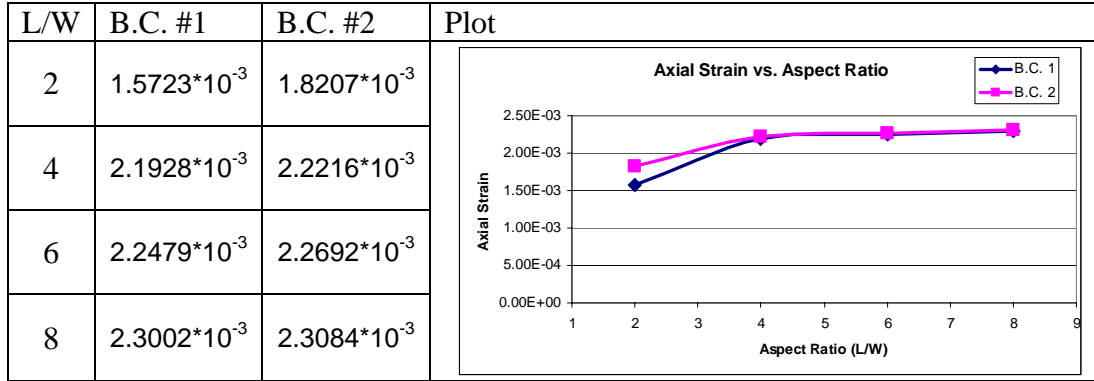
Although both cases converge as L/W becomes larger, changes in the axial strain values form B.C. 1 are larger. The values become stable as the L/W increased to 4 or higher. The result from the CLT method is $2.578*10^{-3}$ which is slightly off from the ANSYS result.

Transverse Strains (εY)
    Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	-0.926E-03	-0.972E-03	
4	-1.335E-03	-1.216E-03	
6	-1.118E-03	-1.096E-03	
8	-1.053E-03	-1.042E-03	
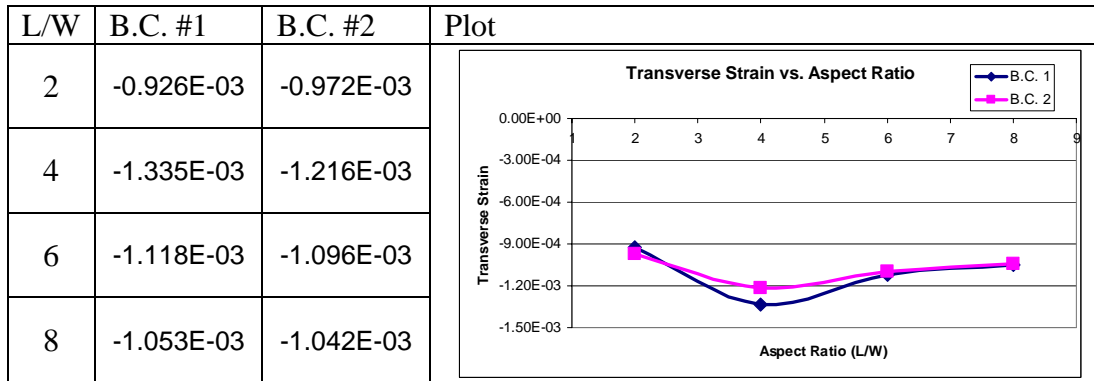
The strain values are approximately $-1.05*10^{-3}$. The values for both cases are gradually shifting to the neighborhood of this value while the difference in the values between the two boundary conditions is decreasing. $-0.941*10^{-3}$ is the CLT result, so the values are matching.

XX

Plots of Shear Strains (γXY) measured along the length

Shear strain values form the ANSYS results are plotted as shown.
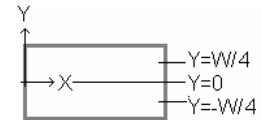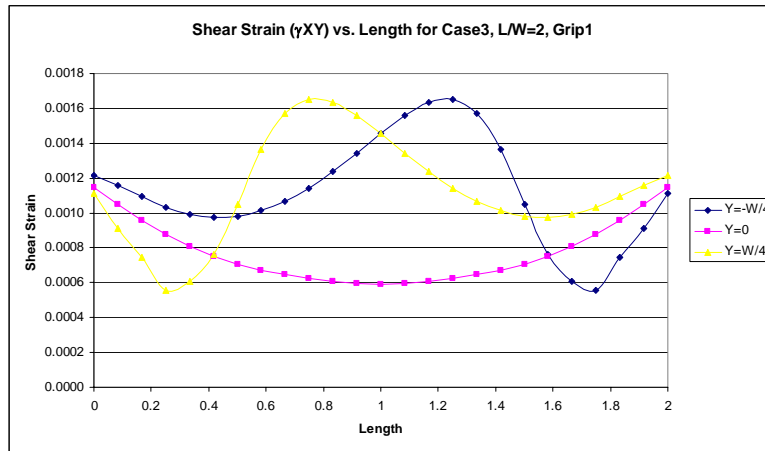
The Shear values are taken along the length at Y=W/4. Y=0, Y=-W/4. The values at Y=W/4 and Y=-W/4 are mirror images, while the values determined at Y=0 are a symmetric with zero slope at the middle. The shear stresses at Y=0 decreases close to the center due to the boundary condition.
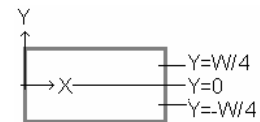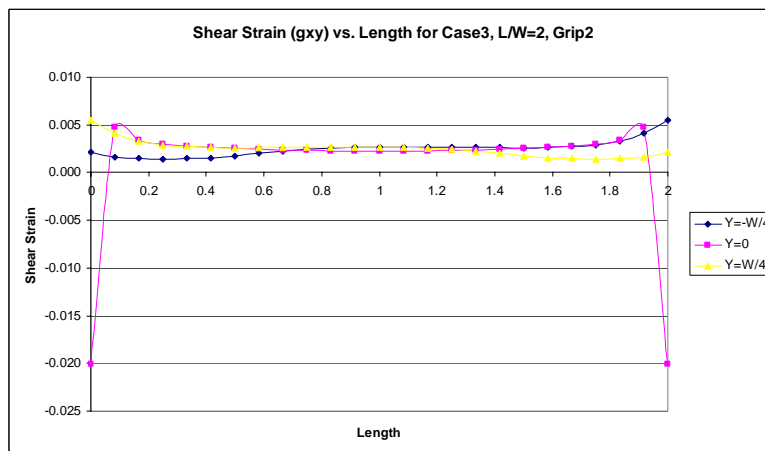
Except at X=0 and X=L, the shear strain values are generally the same at all places.

Shear strain (γXY)
    Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	0.5914E-03	2.2150E-03	
4	3.1285E-03	3.3823E-03	
6	3.6853E-03	3.8014E-03	
8	3.9828E-03	4.0319E-03	
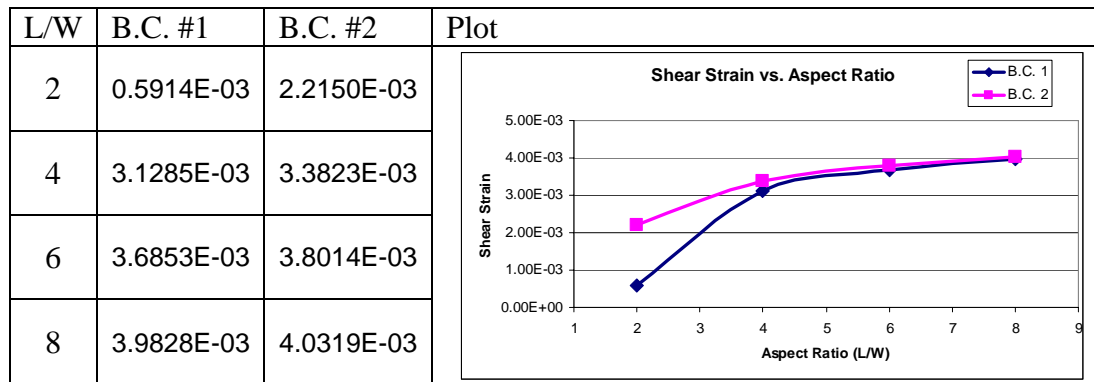
The shear strain value converges as L/W becomes larger. The difference between the BC1 and BC2 results gets smaller. However, the CLT result is $4.933*10^{-3}$, meaning that ANSYS results will not be matching with the CLT.

Case 4: Multi-layer Biased Case

Plot of Axial Strains (εx) along the length
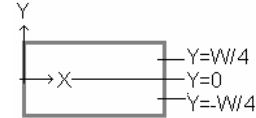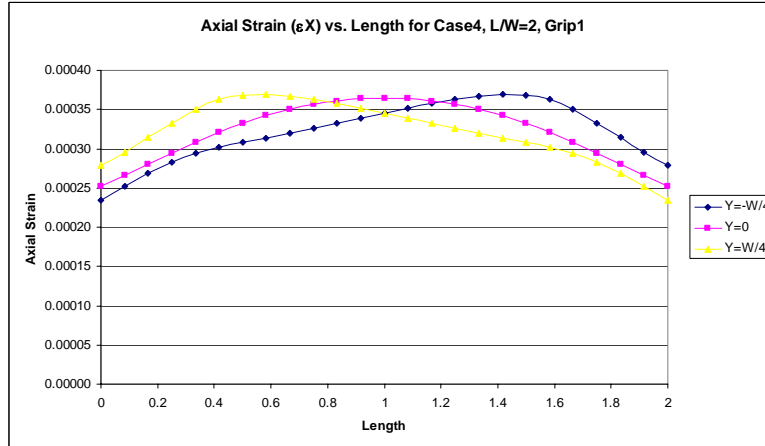
The axial strain distribution for the Y=0 is maximum at the center while the slope slowly decreases. The curves for the Y=W/4, and Y=-W/4 have maximum values at X=3L/4 and L/4. Both curves follow the trend near the ends of the coupon.
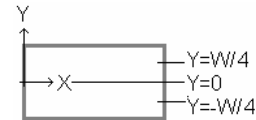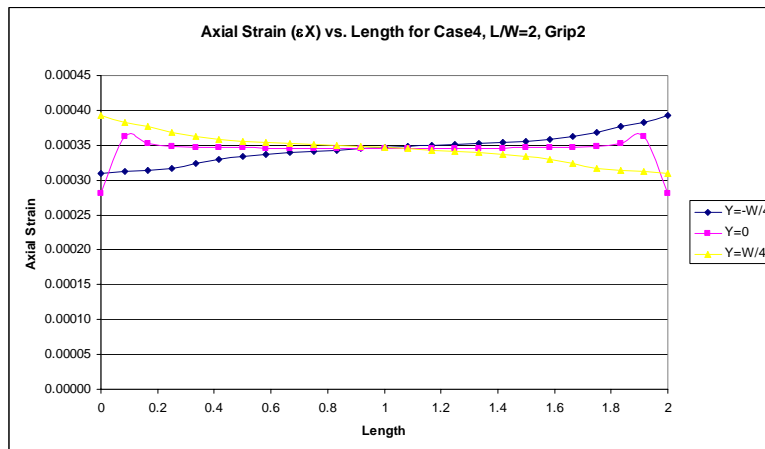
When the axial strain is measured at either Y=W/4 or –W/4, one of the ends will have a value of about 0.00040 and the other end will be around 0.00031. In comparison, the value taken at Y=0 is relatively uniform and stays around 0.00035 except at the end. That's due to the boundary conditions applied.

Axial Strains (εx)

      Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	3.6459E-04	3.4566E-04	
4	3.5192E-04	3.4948E-04	
6	3.4963E-04	3.5025E-04	
8	3.5052E-04	3.5065E-04	

Axial Strain vs. Aspect Ratio
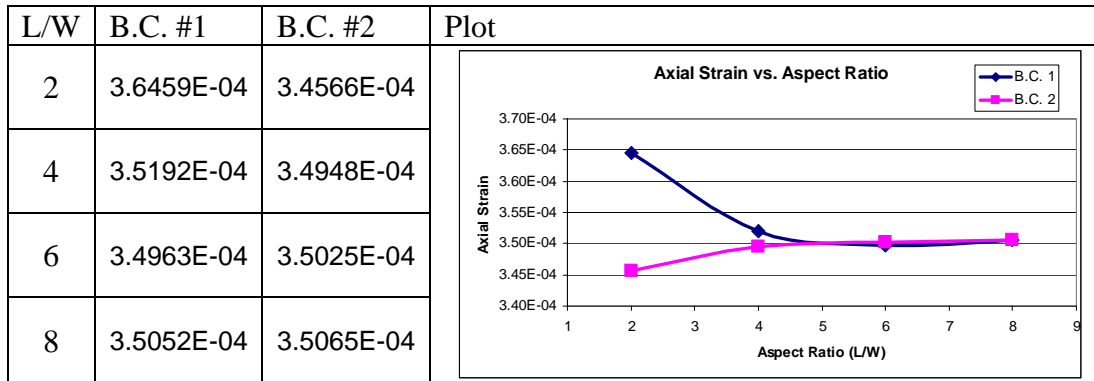
The axial strain values will stabilize as the aspect ratio becomes larger.
The CLT result indicates $3.521*10^{-4}$, meaning they are a close match after the L/W of 6 or higher.

Transverse Strains (εY)

      Measured at Center X=L/2, Y=0

L/W	B.C. #1	B.C. #2	Plot
2	-4.002E-04	-3.904E-04	
4	-4.107E-04	-3.951E-04	
6	-3.934E-04	-3.933E-04	
8	-3.922E-04	-3.920E-04	

Transverse Strain vs. Aspect Ratio
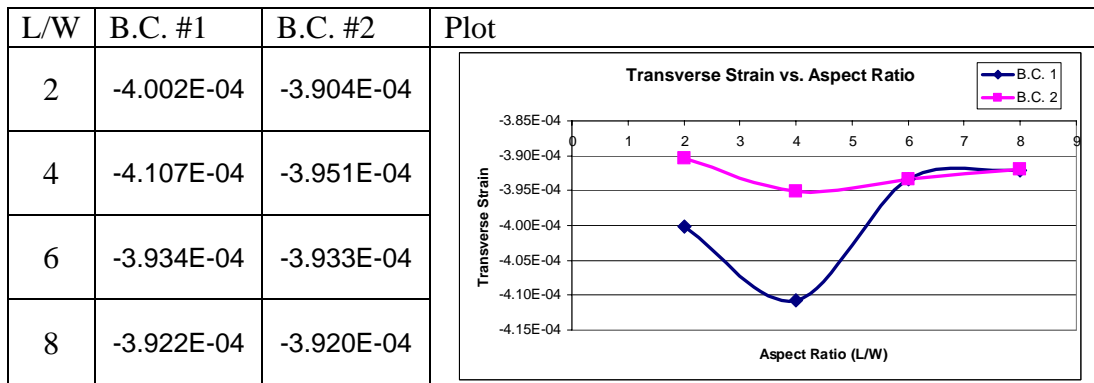
There is a slight decrease in B.C #1 at L/W=4 but the curves are following the trend of convergence as the L/W increase. $-3.900*10^{-4}$ is the CLT result, which are similar values from the both boundary conditions.

Plot of Shear Strains (γXY) along the length
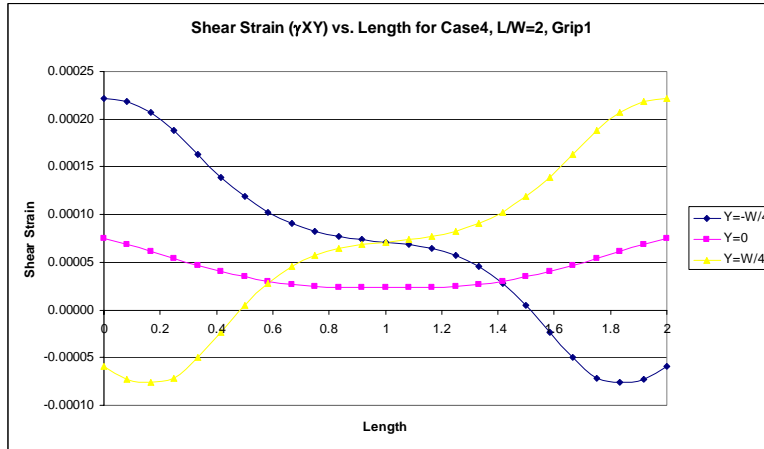
Y=W/4 and Y=-W/4 are mirror images and the rate of change becomes very slow at the center and at the ends. The values taken at Y=0 gradually decrease.
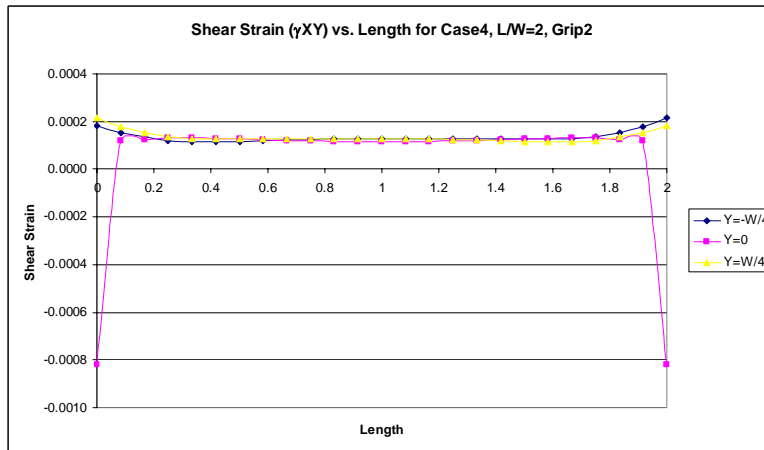
As seen for Case 3, the values at Y = 0 have a very large divergence near the end points. Except that portion, all three curves are very smooth with not much change, indicating that the shear strain values are almost uniform within the boundary.

<u>Shear Strain (γXY)</u>

The shear strain values are taken at the center of the specimens.

L/W	B.C. #1	B.C. #2	Plot
2	0.2334E-04	1.1491E-04	
4	1.1967E-04	1.3608E-04	
6	1.5282E-04	1.5941E-04	
8	1.6859E-04	1.7161E-04	

Shear Strain vs. Aspect Ratio

Shear Strain vs. Aspect Ratio chart with legend B.C. 1, B.C. 2; Y-axis Shear Strain from 0.00E+00 to 2.00E-04; X-axis Aspect Ratio (L/W) from 1 to 9.
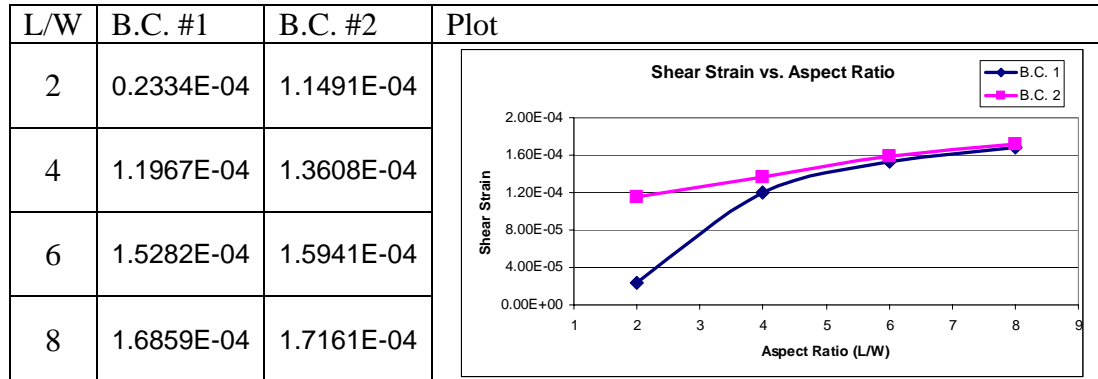
This plot is similar to the axial strain case. The value for B.C. 1 converges quickly, while the value for B.C.2 gradually increases. Although both cases have a big difference in the values at lower aspect ratio, the values become very similar as the effect of boundary conditions diminishes. Comparing to the value estimate from the CLT method was $1.972*10^{-4}$ and a little off from the ANSYS results, but the change in the strain value is positive so the result from CLT supports the values from ANSYS.

Results from Case 5: Balanced Laminate with κ

Establishing the Basic Concept and Verifying the Result

The function of Classical Lamination Theory (CLT) is not only to observe the force and strain value relationship, but From CLT, the coupon could have curvature after loading due to unsymmetiric layup. For example, consider a laminate made of one layer of carbon fiber at 0 degrees on top of 90derees. The layer with 0 degree fiber direction has a much higher modulus value than others, causing the layers to elongate unevenly and result in a curved coupon.
Since this study is based on CLT theory, delamination is ignored.

Case 5: Laminate with κ in Linear and Rigid Grips

The case with one free end will have curvature, while the ends with constrains will develop a concaved specimen during the loading.
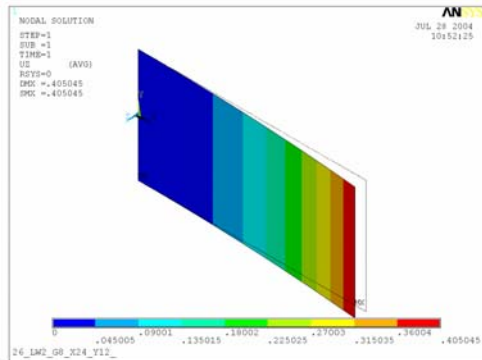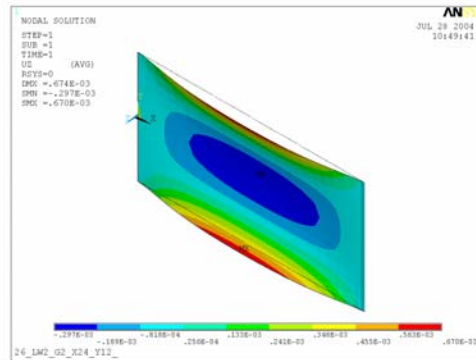
One free end   Both ends constrained

The boundary condition #2 has constraints in vertical direction (Z) for both ends. While the boundary condition #3 has one free end, which allow the specimen to have vertical displacement.
The values from two different boundary conditions result in completely different results therefore, the values were not compared.

Axial Strain (εX) measured at center (X=L/2, Y=0)

L/W	B.C. 2	B.C. 3
2	6.158E-04	-3.880E-04
4	6.183E-04	-3.880E-04
6	6.182E-04	-3.880E-04
8	6.181E-04	-3.880E-04

Axial strain is constant for all aspect ratios. The values observed in the B.C. 2 are positive since the specimen in being stretched from the applied load. The strain value calculated form the CLT method is $6.156*10^{-4}$ and this is closely matching with the result in this case. The result from the B.C. 3 indicate a negative value since the strain is measured at the top of the laminate and the tip of the coupon is lifted, causing the top surface to have negative value while the bottom surface has positive strain value.

Transverse Strain (εY) measured at center (X=L/2, Y=0)

L/W	B.C. 2	B.C. 3
2	-1.201E-04	-5.543E-05
4	-1.267E-04	-5.543E-05
6	-1.266E-04	-5.543E-05
8	-1.266E-04	-5.543E-05

The result from the CLT method shows $-2.486*10^{-5}$ as the result but the values from ANSYS solution show no match with value regardless of the boundary condition.

Shear Strain (γXY) measured at center (X=L/2, Y=0)

L/W	B.C. 2	B.C. 3
2	-6.874E-17	-2.579E-13
4	3.935E-14	-1.168E-12
6	1.282E-13	-4.438E-12
8	2.373E-13	-8.735E-12

CLT=0 and values observed form ANSYS result are zero.

Summary

The boundary conditions made differences in result values for all kinds of strain values. The case using boundary condition 2 shows the result values close to the values from the CLT method except for the transverse strain. Since the B.C. #2 is one of two conditions used in this study, the strain values acquired from the ANSYS should be similar to what is calculated in the CLT method even the coupon has unsymmetric laminate schedule.

Study of Coupling Effect Using ANSYS

Establishing the Basic Concept and Verifying the Result

In reference #2, a coupon underwent a series of tensile tests to study the characteristics of the laminate with coupling. This section is to confirm that ANSYS is capable of dealing with laminates with tensile-twist coupling.

Basic layup and dimensions:

Reference #2 used an 8 layered configuration. Although there was no specific thickness, 0.000125m was selected as the initial value. The value picked was based on the notes given in Basic Composite Material class (AE653) for a typical prepreg material. The offset angle was set to be 30 degrees as stated. After trying with several different thicknesses, the thickness value of 0.000175m matches the reference 2 result.

Twist rate and load comparison:

A chart was made to present the relationships between the magnitude of the load against rotation and displacement.
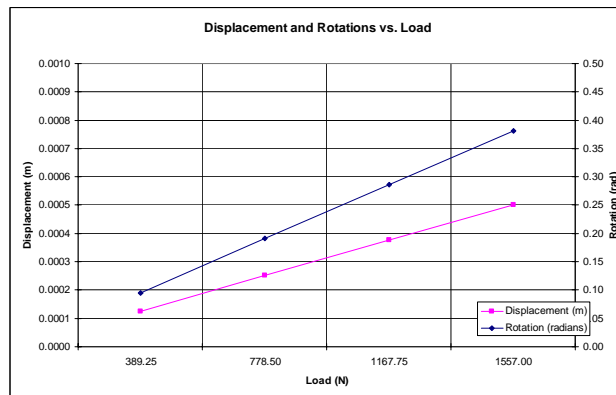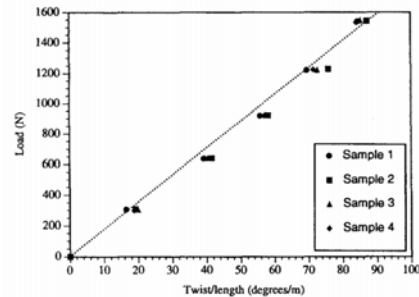
ANSYS result   Plot from Ref #2

From the result, the both displacements and twists behave linearly to the magnitude of the load applied. Therefore the ANSYS can provide linear elastic model with coupling under a designated boundary condition.

Case 6: Coupled Laminate in Rotating Grip and in Rigid Grips

When a coupon has a coupling, it will be twisted when a tensile load is applied. This is simply due to the internal shear stress caused within the laminate. If both ends have constraints, the amount of twist developed will be much less, and will cause stress concentrations at or near the ends since the grips are resisting the moment created by the laminate. All the values are measured at the center of the specimen (X=L/2, Y=0).

Axial Strain (εX)

L/W	B.C. 2	B.C. 4
2	5.003E-04	8.393E-04
4	5.005E-04	8.623E-04
6	5.004E-04	8.568E-04
8	5.004E-04	8.564E-04

Although the values from the two different boundary conditions do not match, these values do not change at any aspect ratio. From the observation, having a free end increases the amount of strain. The strain value by the CLT method indicates $4.962*10^{-4}$, which is the value closed to the result in the B.C. #2 which is similar to the plane stress model. The plot shows the axial strain for the B.C. 2.
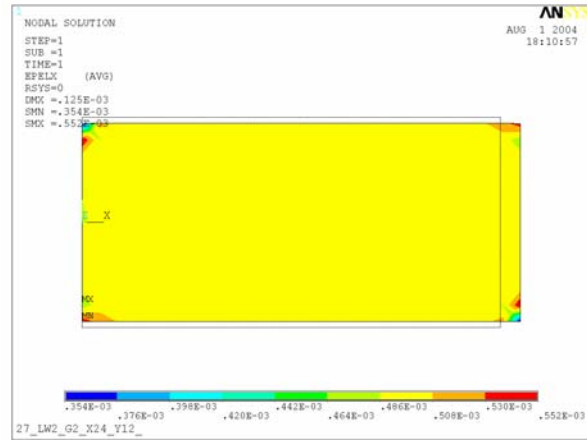
Transverse Strain (εY)

L/W	B.C. 2	B.C. 4
2	-5.932E-04	-4.974E-04
4	-5.939E-04	-5.487E-04
6	-5.939E-04	-5.349E-04
8	-5.938E-04	-5.331E-04

The values from the B.C. 2 do not change as the aspect ratio change but the results from the other condition are largely affected by the aspect ratio. -$5.895*10^{-4}$ is the value from the CLT method and are matching with the result from the B.C. 2. The plot shows the transverse strain for the B.C. 2.
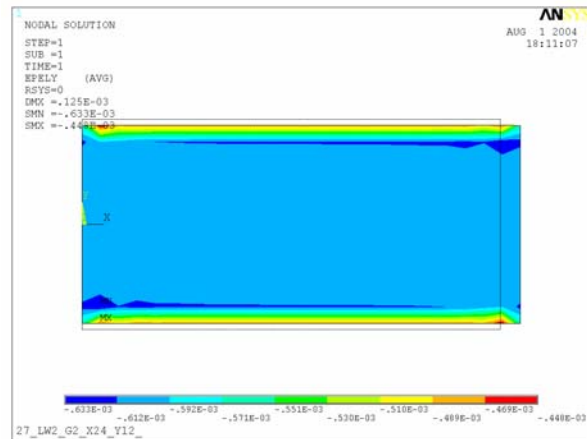
Shear Strain (γXY)

L/W	B.C. 2	B.C. 4
2	5.349E-07	-2.139E-03
4	1.676E-07	-2.294E-03
6	3.282E-07	-2.297E-03
8	4.258E-07	-2.296E-03

The shear strain for this case for B.C. #4 shows notable amount of strain, while the values from the B.C. #2 is small. The strain value determined from the CLT is 0. The plot shows the shear strain for the B.C. 2.
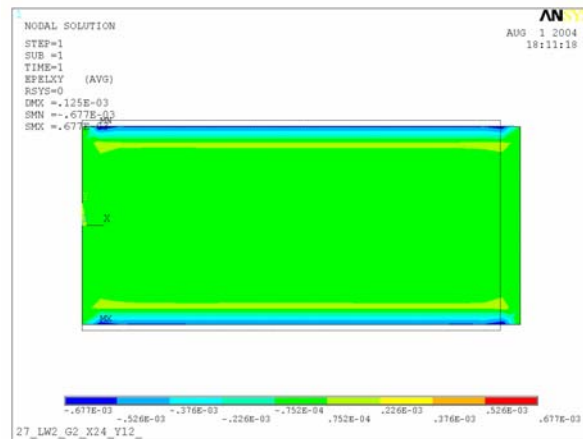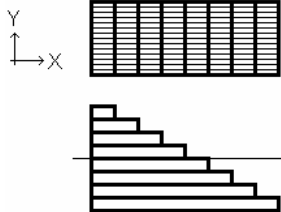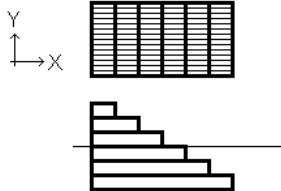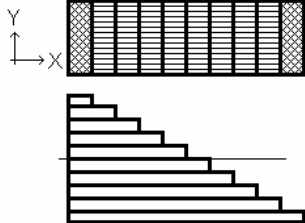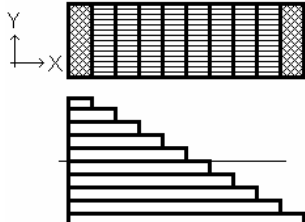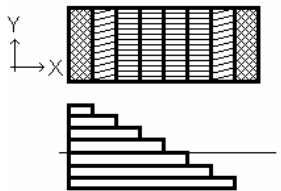
Summary
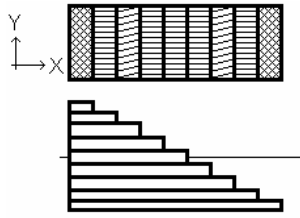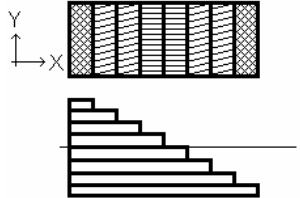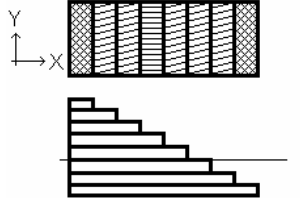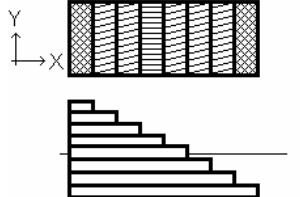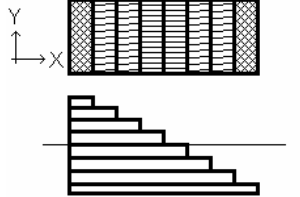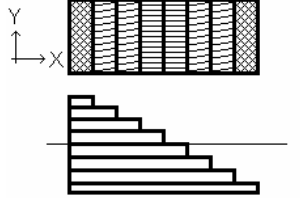
If a boundary conditions which constraint the rotation of the specimen, the result becomes very different from the value determined from the CLT method. However, the values acquired from the result of boundary condition 2, which restricts the movement in Z direction at the both ends will have result similar to the CLT. In addition, the changes in the strain values are small and can be said for the all three strain components. The strain plots support this idea.
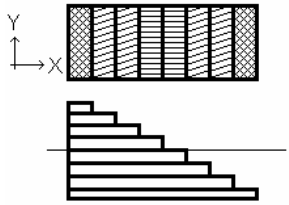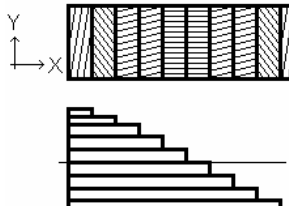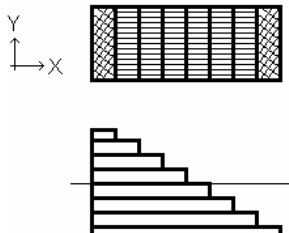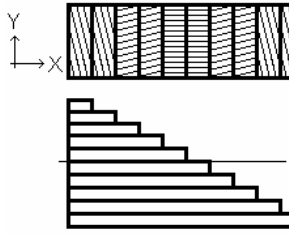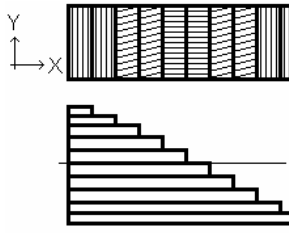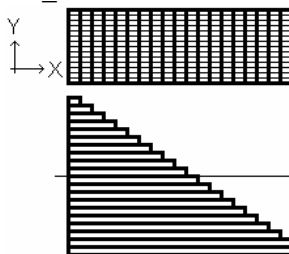
Appendix C: Results of Proposed Cases
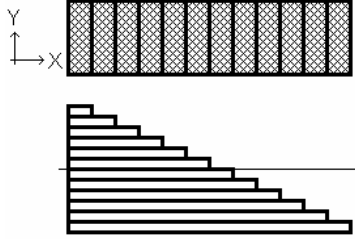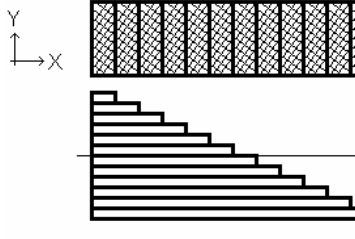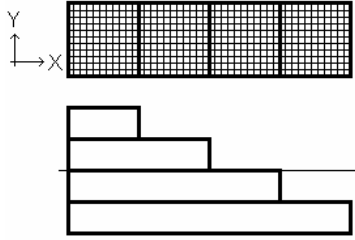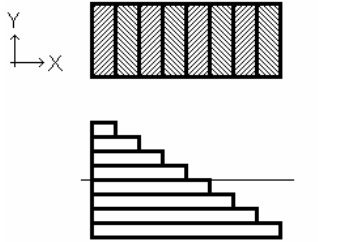
Laminate Schedules of the All Proposed Cases

These are the list and the descriptions of specimens analyzed.

Case#	Configuration	Laminate Description
1	SC_GDU16	Glass at 0°, 0.0004075m (8) Total thickness: 0.0032600m
2	SC_CUD	Glass at 0°, 0.0005183m (6) Total thickness: 0.003198m
3	85GUD_15GDB	Biax at 0°, 0.000280m (1) Glass at 0°, 0.0004075m (6) Biax at 0°, 0.000280m (1) Total thickness: 0.00382m
4	85CUD_15GDB	Biax at 0°, 0.000280m (1) Carbon at 0°, 0.0005183m (6) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m
5	55CUD_30C20_15GDB	Biax at 0°, 0.000280m (1) Carbon at 20°, 0.0005183m (1) Carbon at 0°, 0.0005183m (4) Carbon at 20°, 0.0005183m (1) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m

Case#	Configuration	Laminate Description
6	55GUD_30C20_15GDB	Biax at 0°, 0.000280m (1) Glass at 0°, 0.0004075m (1) Carbon at 20°, 0.0005183m (1) Glass at 0°, 0.0004075m (3) Carbon at 20°, 0.0005183m (1) Glass at 0°, 0.0004075m (1) Biax at 0°, 0.000280m (1) Total thickness: 0.0036341m
7	30CUD_55C20_15GDB	Biax at 0°, 0.000280m (1) Carbon at 20°, 0.0005183m (2) Carbon at 0°, 0.0005183m (2) Carbon at 20 °, 0.0005183m (2) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m
8	15CUD_70C20_15GDB	Biax at 0°, 0.000280m (1) Carbon at 20°, 0.0005183m (2) Carbon at 0°, 0.0005183m (1) Carbon at 20°, 0.0005183m (3) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m
9	10GUD_75C20_15GDB	Biax at 0°, 0.000280m (1) Carbon at 20°, 0.0005183m (2) Glass at 0°, 0.0004075m (1) Carbon at 20°, 0.0005183m (3) Biax at 0°, 0.000280m (1) Total thickness: 0.0035590m
10	30CUD_55C10_15GDB	Biax at 0°, 0.000280m (1) Carbon at 10°, 0.0005183m (2) Carbon at 0°, 0.0005183m (2) Carbon at 10°, 0.0005183m (2) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m
11	30CUD_55C15_15GDB	Biax at 0°, 0.000280m (1) Carbon at 15°, 0.0005183m (2) Carbon at 0°, 0.0005183m (2) Carbon at 15° deg, 0.0005183m (2) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m

Case#	Configuration	Laminate Description
12	30CUD_55C25_15GDB	Biax at 0°, 0.000280m (1) Carbon at 25°, 0.0005183m (2) Carbon at 0°, 0.0005183m (2) Carbon at 25°, 0.0005183m (2) Biax at 0°, 0.000280m (1) Total thickness: 0.0036698m
13	30CUD_55C20_15G75-55	Glass at 75°, 0.000170m (1) Glass at -55°, 0.0001700m (1) Carbon at 20°, 0.0005175m (2) Carbon at 0°, 0.0005175m (2) Carbon at 20°, 0.0005175m (2) Glass at -55°, 0.0001700m (1) Glass at 75°, 0.0001700m (1) Total thickness: 0.0037850m
14	85CUD_15G20-70	Biax at -25°, 0.000280m (1) Carbon at 0°, 0.0005183m (6) Biax at -25°, 0.000280m (1) Total thickness: 0.0036698m
15	30CUD_55C20_15G-70	Glass at -70°, 0.0004100m (2) Carbon at 20°, 0.0005175m (2) Carbon at 0°, 0.0005175m (2) Carbon at 20°, 0.0005175m (2) Glass at -70°, 0.0004100m (2) Total thickness: 0.0047500m
16	30CUD_55C20_15G90	Glass at 90°, 0.0001715m (2) Carbon at 20°, 0.0005175m (2) Carbon at 0°, 0.0005200m (2) Carbon at 20°, 0.0005175m (2) Glass at 90°, 0.0001715m (2) Total thickness: 0.0037960m
17	SC_GUD06	Glass at 0°, 0.0001715m (20) Total thickness: 0.0034300m

Case#	Configuration	Laminate Description
18	SK_GDB12	Biax at 0°, 0.0002800m (12) Total thickness: 0.0033600m
19	SK_G20-70	Biax at -25°, 0.0002800m (12) Total thickness: 0.0033600m
20	SK_GDB38	Biax at 45°, 0.0009000m (1) Biax at -45°, 0.0009000m (1) Biax at -45°, 0.0009000m (1) Biax at 45°, 0.0009000m (1) Total thickness: 0.003600m
21	SK_CDB	Carbon at +45°, 0.0005175m (1) Carbon at -45°, 0.0005175m (1) Carbon at +45°, 0.0005175m (1) Carbon at -45°, 0.0005175m (1) Carbon at -45°, 0.0005175m (1) Carbon at +45°, 0.0005175m (1) Carbon at -45°, 0.0005175m (1) Carbon at +45°, 0.0005175m (1) Total thickness: 0.004140m

Classification of the Laminates

Case#	Title	Balanced/ Biased	Symmetirc/ Unsymmetric
1	SC_GDU16	Balanced	Symmetric
2	SC_CUD	Balanced	Symmetric
3	85GUD_15GDB	Balanced	Symmetric
4	85CUD_15GDB	Balanced	Symmetric
5	55CUD_30C20_15GDB	Biased	Symmetric
6	55GUD_30C20_15GDB	Biased	Symmetric
7	30CUD_55C20_15GDB	Biased	Symmetric
8	15CUD_70C20_15GDB	Biased	Unsymmetric
9	10GUD_75C20_15GDB	Biased	Unsymmetric
10	30CUD_55C10_15GDB	Biased	Symmetric
11	30CUD_55C15_15GDB	Biased	Symmetric
12	30CUD_55C25_15GDB	Biased	Symmetric
13	30CUD_55C20_15G75-55	Biased	Symmetric
14	85CUD_15G20-70	Biased	Symmetric
15	30CUD_55C20_15G-70	Biased	Symmetric
16	30CUD_55C20_15G90	Biased	Symmetric
17	SC_GUD06	Balanced	Symmetric
18	SK_GDB12	Balanced	Symmetric
19	SK_G20-70	Biased	Symmetric
20	SK_GDB38	Balanced	Symmetric
21	SK_CDB	Balanced	Symmetric

Strain Results Measured at Center from the Each Case:

Axial Strain (L/W=8, W=0.6m, Load=1200N)

Case#	Title	B.C. #1	B.C. #2
1	SC_GDU16	7.727E-05	7.727E-05
2	SC_CUD	2.684E-05	2.684E-05
3	85GUD_15GDB	7.388E-05	7.388E-05
4	85CUD_15GDB	2.642E-05	2.642E-05
5	55CUD_30C20_15GDB	3.147E-05	3.149E-05
6	55GUD_30C20_15GDB	6.740E-05	6.744E-05
7	30CUD_55C20_15GDB	4.356E-05	4.362E-05
8	15CUD_70C20_15GDB	5.519E-05	5.538E-05
9	10GUD_75C20_15GDB	7.542E-05	7.562E-05
10	30CUD_55C10_15GDB	3.189E-05	3.192E-05
11	30CUD_55C15_15GDB	3.749E-05	3.754E-05
12	30CUD_55C25_15GDB	4.927E-05	4.933E-05
13	30CUD_55C20_15G75-55	4.261E-05	4.266E-05
14	85CUD_15G20-70	2.616E-05	2.616E-05
15	30CUD_55C20_15G-70	4.071E-05	4.075E-05
16	30CUD_55C20_15G90	4.379E-05	4.385E-05
17	SC_GUD06	7.344E-05	7.344E-05
18	SK_GDB12	3.242E-04	3.242E-04
19	SK_G20-70	2.113E-04	2.114E-04
20	SK_GDB38	1.250E-04	1.250E-04
21	SK_CDB	1.472E-04	1.472E-04

Transverse Strain (L/W=8, W=0.6m, Load=1200N)

Case#	Title	B.C. #1	B.C. #2
1	SC_GDU16	-2.241E-05	-2.241E-05
2	SC_CUD	-7.519E-06	-7.516E-06
3	85GUD_15GDB	-2.750E-05	-2.750E-05
4	85CUD_15GDB	-9.556E-06	-9.553E-06
5	55CUD_30C20_15GDB	-1.681E-05	-1.675E-05
6	55GUD_30C20_15GDB	-3.619E-05	-3.613E-05
7	30CUD_55C20_15GDB	-2.532E-05	-2.518E-05
8	15CUD_70C20_15GDB	-3.136E-05	-3.119E-05
9	10GUD_75C20_15GDB	-4.276E-05	-4.255E-05
10	30CUD_55C10_15GDB	-1.493E-05	-1.490E-05
11	30CUD_55C15_15GDB	-2.018E-05	-2.010E-05
12	30CUD_55C25_15GDB	-2.936E-05	-2.916E-05
13	30CUD_55C20_15G75-55	-1.915E-05	-1.907E-05
14	85CUD_15G20-70	-7.507E-06	-7.507E-06
15	30CUD_55C20_15G-70	-2.003E-05	-2.007E-05
16	30CUD_55C20_15G90	-1.387E-05	-1.379E-05
17	SC_GUD06	-2.130E-05	-2.130E-05
18	SK_GDB12	-2.065E-04	-2.065E-04
19	SK_G20-70	-9.210E-05	-9.228E-05
20	SK_GDB38	-1.516E-05	-1.516E-05
21	SK_CDB	-1.114E-04	-1.114E-04

Shear Strain (L/W=8, W=0.6m, Load=1200N)

Case#	Title	B.C. #1	B.C. #2
1	SC_GDU16	-7.295E-17	-1.034E-16
2	SC_CUD	-2.318E-16	-3.363E-16
3	85GUD_15GDB	-1.647E-16	-2.737E-16
4	85CUD_15GDB	-9.079E-17	-1.515E-16
5	55CUD_30C20_15GDB	-2.328E-05	-2.375E-05
6	55GUD_30C20_15GDB	-5.758E-05	-5.823E-05
7	30CUD_55C20_15GDB	-5.199E-05	-5.281E-05
8	15CUD_70C20_15GDB	-7.535E-05	-7.659E-05
9	10GUD_75C20_15GDB	-1.070E-04	-1.084E-04
10	30CUD_55C10_15GDB	-3.346E-05	-3.420E-05
11	30CUD_55C15_15GDB	-4.586E-05	-4.671E-05
12	30CUD_55C25_15GDB	-5.207E-05	-5.277E-05
13	30CUD_55C20_15G75-55	-5.232E-05	-5.288E-05
14	85CUD_15G20-70	-2.371E-06	-2.425E-06
15	30CUD_55C20_15G-70	-2.371E-06	-2.425E-06
16	30CUD_55C20_15G90	-5.784E-05	-5.853E-05
17	SC_GUD06	-9.355E-17	-2.147E-16
18	SK_GDB12	-4.605E-16	-6.330E-16
19	SK_G20-70	-1.740E-04	-1.748E-04
20	SK_GDB38	-1.740E-04	-1.748E-04
21	SK_CDB	-7.509E-16	-8.390E-16

Appendix D: ANSYS

ANSYS

ANSYS is a FEA (finite element analysis) program used in many fields of industry. The software performs not only simple static analysis, but some complicated cases such as non-linear bucking problems. In addition, ANSYS can be used in many types of problems including structural, thermodynamic, fluid, and magnetism.

Selection of Element Type

There are more than 10 different kinds of Shell element types available in ANSYS library and each element type has unique purpose. Other possible element types available for this kind of study were Plane182, Solid46, and Solid191. The list of all available shell element is Shell 28, 41, 43, 51, 57, 61, 63, 91, 93, 99, 143, 150, 157, 163, 181. From the initial investigation, the Shell element types can be used are Shell 43, 63, 91, 93, 99 and 181.

Element Type	# of nodes	DOF /node	Special Function / Comments
Shell43 – 4 node plastic large strain shell	4	6	Warped, Moderately-thick shell Plasticity, Creep. Stress stiffening, Large deflection, Large strain
Shell63 – Elastic Shell	4	6	Bending, Membrane capabilities, In-plane and normal load allowed, Stress stiffening, Large deflection
Shell91 – Nonlinear layered Shell	8	6	Layered applications up to 100 layers, Linear or Non-linear material property, Sandwich panel option. Quadratic deformation shape, Plasticity, Stress stiffening, Large deflections, Large strain
Shell93 – 8 Node Structural Shell	8	6	Suitable for curved shell, Plasticity, Stress stiffening, Large deflection, Large strain, Quadratic deformation shape
Shell99 – Linear Layered Structural Shell	8	6	Layered up to 250 layers, Quadratic deformation shape, Linear material properties only
Shell181 – Finite Strain Layered Shell	4	6	Moderately-thick structure, Linear material properties, Large rotation, Larger strain, Linear and Non-linear application, Sandwich panel option.

This study does not require any special function. Therefore, either Shell 91 or Shell 99 is thought to be the most adequate. Both element types are the same, except the Shell 91 was capable of accepting non-linear behavior. From this fact, the Shell 99 element type is chosen.

Consideration Taken for the Shell 99 Element Type

The Shell 99 element type uses 8 nodes for a rectangular element. Different from the 4 node element system, this allows quadratic iteration of the result. The figure on the left shows 4 node element and right one is the 8 node element.

x_{IJ} = Element x-axis if ESYS is not supplied.
x = Element x-axis if ESYS is supplied.

x_{IJ} = Element x-axis if ESYS is not supplied.
x = Element x-axis if ESYS is supplied.
LN = Layer Number
NL = Total Number of Layers
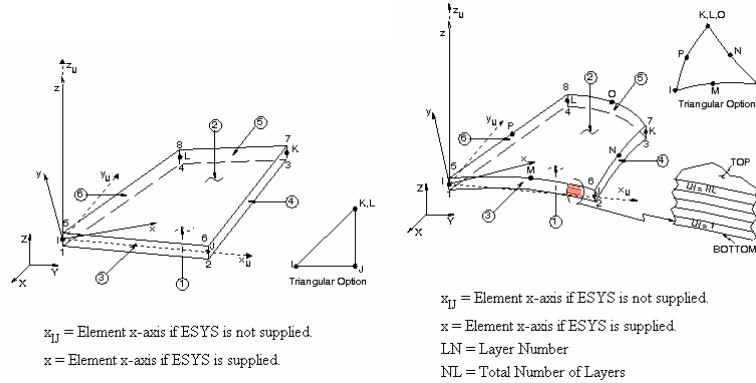
When the layup is being set, the layer is built from the bottom to the top surface. Therefore the user has to pay attention when they are setting up laminate schedules.
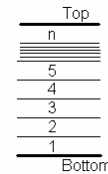
In addition, the normal direction of the elements is defined in a manner that follows the right hand rule.
In addition, the axial fiber direction is the first vector used to define a shell. The elements are defined in such manner that (i, j, k, l, m, n, o, and p) and the direction from i to j is the original primary axis.
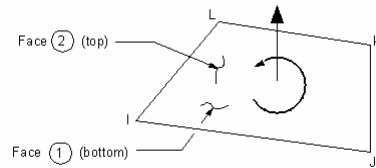
For certain shell elements, the location of the nodes can be set at top, middle or bottom of the element. The default is set at the middle and it will not be changed during the experiment since the mechanical grips position the center of coupon at the middle of the testing machine thus this is the right setting also for the testing on the FEA.

Result Processing

Results can be retrieved either numerically or graphically. The numerical result can be useful when exact value at particular location is needed. This helps determining the rate of convergence, and maximum or minimum values of the model

On the other hand, the graphical plot is useful when identifying the distribution of stresses and strains. Also this can show the location of stress concentration, or discontinuity.

Numerical Results

The basic format of the ANSYS result files includes the case name, function, raw data and the maximum and minimum values for each component at the end. There will be seven components given for each line of the data; node number and six components of the results. The components change as the type of result requested changes.

- Displacements (PRNSOL, DOF) – DX, DY, DY, ROTX, ROTY and ROTZ
- Stresses (PRNSOL, S) – σx, σy, σz, τxy, τyz and τx
- Strains (PRNSOL, EPEL) – εx, εy, εz, γxy, γyz and γxz

The raw data produced from ANSYS results are given in a systematic order. Therefore a macro code created on Excel can process raw data organized in a spreadsheet, then saved as a file in the Microsoft Excel format so that the user can access the result directly later on.

The sample date sheet seen as below
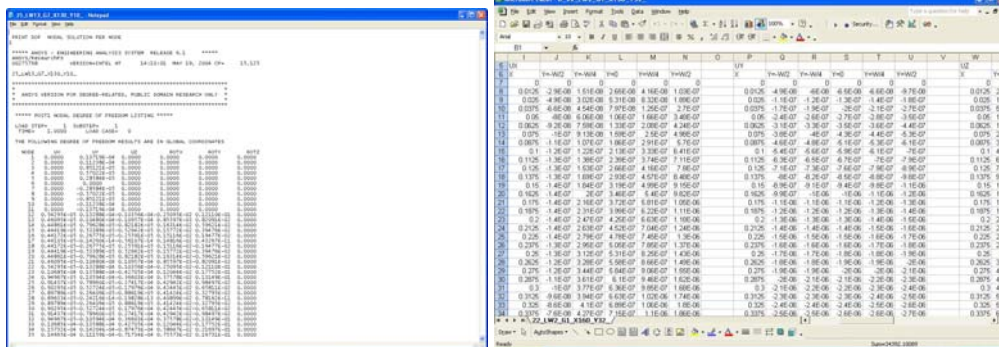
Before (in Memo Pad)   After (In Excel)

The program code is available in later section

Graphical
Available File Types:

The other form of results output is the graphical plot. All the components, displacements, stresses and strains, can be shown in common picture file format.

	Color	Gray Scale	Monochrome
BMP	✓	✗	✗
EPS (post script)	✓	✓	✓
TIFF	✓	✗	✓
JPEG	✓	✓	✓
PNG	✓	✗	✓

Note that the post script files cannot be read unless processed in Adobe Photoshop or other programs supporting EPS files before contour plot can be seen. The components of the result will be the same as the numerical result files. Unlike the numerical result, each component has to output separately. The results can be acquired from nodal results or element results.

Color Format:

The color mode can be selected one of three ways. The figures shown below are an example of the picture in different color mode.
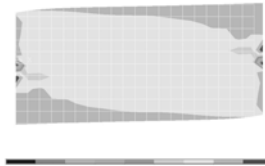
Monochrome (B/W)   Gray Scale   Color
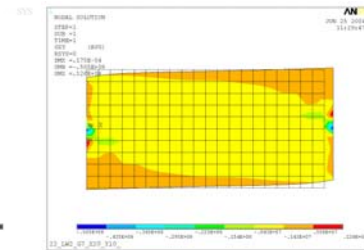
Background Color

The background color is initially set to be black but can be reversed.
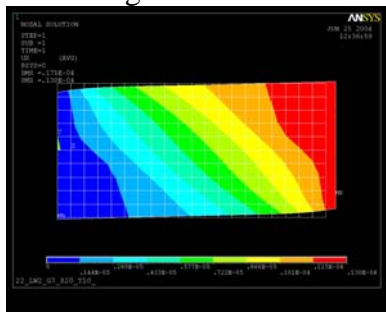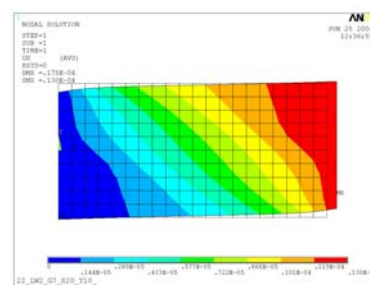
Default output   Reversed mode

Batch Mode and Macro Code Used in Excel

Batch Mode

All the cases were processed using the ANSYS batch mode. Unlike the interactive mode, the batch mode can automatically process multiple cases without reloading the file manually. The other advantages of using the batch mode are reduced processing time and use of less disk space and memory because the application is not required to provide graphical representation. The input codes generated by the Excel's macro function were manually transferred onto Memo Pad in text file format, then directly fed in ANSYS through the batch mode menu.

Code Generation Using Excel Macro

The commands to perform analysis are generated by the macro code in this section. The user sets the case number, aspect ratio, width, element size in X and Y direction, load applied, boundary condition, and condition of tab if required.

```
Sub MakeANSYSCode()
Application.ScreenUpdating = False ' Turn off screen update for faster process

' Clearing the spreadsheet ----------------------------------------
    Sheets("Code").Select
    Range("A1:IV65536").Select
    Selection.ClearContents

For Row = 2 To 280   'Go thtough cases

    If Sheets("Control").Cells(Row, 4) = "" Then ' stop when the Case
        End
    Else

    ' Variables ----------------------------------------------------------
        CRow = 1             ' The initial row number for the code
        CClm = Row - 1       ' Code Column in the code sheet

        MClm = 5             ' Column # for Mat# in LYR Sheet
        AClm = 6             ' Column # for Angle in LYR Sheet
        TClm = 7             ' Column # for Thickness in LYR Sheet

        MatMAX = 5           ' Max # of Materials

        C = Sheets("Control").Cells(Row, 4)     ' Case #
        W = Sheets("Control").Cells(Row, 5)     ' Basic Width
        LWR = Sheets("Control").Cells(Row, 6)   ' Length/Width Ratio
        ESX = Sheets("Control").Cells(Row, 7)   ' Elem Size in X direction
        ESY = Sheets("Control").Cells(Row, 8)   ' Elem Size in Y direction
        ETYP = Sheets("Control").Cells(Row, 9)  ' Element Type
        ATYP = Sheets("Control").Cells(Row, 10) ' Analysis type
        LD = Sheets("Control").Cells(Row, 11)   ' Load
        GTYP = Sheets("Control").Cells(Row, 12) ' Grip Type (1,2,3,4,5)
        PSS = Sheets("Control").Cells(Row, 13)  ' Switch for Plane Stress
        LEnd = Sheets("Control").Cells(Row, 14) ' Element Length at Ends
        TEnd = Sheets("Control").Cells(Row, 15) ' Element Thickness at Ends

        If TEnd = 0 Then
            MsgBox ("Thickness cannot be 0")
            End
        End If

        NEXE = Int(LEnd / ESX)                  ' Number of element in X at End

        NEX = Int(W * LWR / ESX) + 2 * NEXE     ' # of element in X direction
        NEY = Int(W / ESY)                      ' # of element in Y direction

    ' Deciding the amount of force and direction -------------------------
        If AType = "Tensile" Then
```

```vba
        LD = Round(LD, 0)
    ElseIf AType = "Compression" Then
        LD = Round(-LD, 0)
    End If

' Determinng the Analysis type -----------------------------------------------
    TNote = "LW" & LWR & "_G" & GTYP & "_X" & NEX & "_Y" & NEY  ' Adding technical notes

    ' For plane stress, add "PS" notation.
    If PSS = "Y" Then
        TNote = TNote & "_PS"
    Else
        TNote = TNote & "_"
    End If

    dx = (W * LWR + ESX * NEXE * 2) / NEX ' DX
    dy = W / NEY          ' DY
    X0 = 0               ' X0
    Y0 = -W / 2          ' Y0

    NMJR = (NEX + 1) * (NEY + 1)    ' # of Major node
    NMNR = (NEX + 1) * (NEY)        ' # of Minor node
    NMID = (NEX) * (NEY + 1)        ' # of mid node

    FNAME = C & "_" & TNote         ' Generating the file name

' Clear Everything ----------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "/CLEAR,NOSTART"
    CRow = CRow + 1
' In the batch mode ----------------------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "BATCH,LIST"
    CRow = CRow + 1
' Change the filename -------------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "/FILNAME," & FNAME & ",1"
    CRow = CRow + 1
' Change the title ---------------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "/TITLE," & FNAME
    CRow = CRow + 1
' Set the Units (SI unit) -------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "/UNITS,SI"
    CRow = CRow + 1
' Move to preprocessor menu ----------------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "/PREP7"
    CRow = CRow + 1

' Enter Material Information
    For Row1 = 2 To MatMAX + 1

        MNum = Sheets("Material").Cells(Row1, 1) ' Set Matetial Number

        EX = "MP,EX," & MNum & "," & Sheets("Material").Cells(Row1, 3)        'EX
            Sheets("Code").Cells(CRow, CClm) = EX
            CRow = CRow + 1
        EY = "MP,EY," & MNum & "," & Sheets("Material").Cells(Row1, 4)        'EY
            Sheets("Code").Cells(CRow, CClm) = EY
            CRow = CRow + 1
        EZ = "MP,EZ," & MNum & "," & Sheets("Material").Cells(Row1, 5)        'EZ
            Sheets("Code").Cells(CRow, CClm) = EZ
            CRow = CRow + 1
        GXY = "MP,GXY," & MNum & "," & Sheets("Material").Cells(Row1, 6)      'GXY
            Sheets("Code").Cells(CRow, CClm) = GXY
            CRow = CRow + 1
        GYZ = "MP,GYZ," & MNum & "," & Sheets("Material").Cells(Row1, 7)      'GYZ
            Sheets("Code").Cells(CRow, CClm) = GYZ
            CRow = CRow + 1
        GXZ = "MP,GXZ," & MNum & "," & Sheets("Material").Cells(Row1, 8)      'GXZ
            Sheets("Code").Cells(CRow, CClm) = GXZ
            CRow = CRow + 1
        DENS = "MP,DENS," & MNum & "," & Sheets("Material").Cells(Row1, 9)    'Dens
            Sheets("Code").Cells(CRow, CClm) = DENS
            CRow = CRow + 1
        PRXY = "MP,PRXY," & MNum & "," & Sheets("Material").Cells(Row1, 10)   'PRXY
            Sheets("Code").Cells(CRow, CClm) = PRXY
            CRow = CRow + 1
        PRYZ = "MP,PRYZ," & MNum & "," & Sheets("Material").Cells(Row1, 11)   'PRYZ
            Sheets("Code").Cells(CRow, CClm) = PRYZ
            CRow = CRow + 1
        PRXZ = "MP,PRXZ," & MNum & "," & Sheets("Material").Cells(Row1, 12)   'PRXZ
            Sheets("Code").Cells(CRow, CClm) = PRXZ
            CRow = CRow + 1
    Next
```

XLIII

```vba
' Look for the matching case number --------------------------------
    For RowLYR = 1 To 1000
        If Sheets("LYR").Cells(RowLYR, 1) = C Then
            NumLYR = Sheets("LYR").Cells(RowLYR, 3)              ' Number of layers
            Row1st = RowLYR                                     ' First row
            RowLast = RowLYR + NumLYR - 1                       ' Last row
            TThick = Sheets("LYR").Cells(RowLYR + NumLYR, 7) ' Total thickness
            Exit For
        End If
    Next

' Set the element type (Shell99 or Shell91)
    If ETYP = "SHELL99" Then
        Sheets("Code").Cells(CRow, CClm) = "ET,1,SHELL99"
        CRow = CRow + 1
    ElseIf ETYP = "SHELL91" Then
        Sheets("Code").Cells(CRow, CClm) = "ET,1,SHELL91,,1"
        CRow = CRow + 1
    End If
        Sheets("Code").Cells(CRow, CClm) = "KEYOPT,1,8,1"
        CRow = CRow + 1

' If the case found, then
    If ETYP = "SHELL99" Then     ' for Shell99 elements

        Sheets("Code").Cells(CRow, CClm) = "R,1," & NumLYR & ",,,,"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "RMORE,,,,,,"
        CRow = CRow + 1

        ' Determine if the number of layers is even or odd
            EvenOdd = NumLYR Mod 2

            If EvenOdd = 0 Then 'even number
                Turns = Int(NumLYR / 2)
            Else ' odd number of layer
                Turns = Int(NumLYR / 2) + 1
            End If

        For Turn = 1 To Turns ' Writing the code for RCN

            R1 = RowLast - 2 * (Turn - 1)
            R2 = R1 - 1

            If Turn = Turns And EvenOdd = 1 Then ' Last layer at odd # of layers
                M1 = Sheets("LYR").Cells(R1, MClm)          ' Material #
                A1 = Sheets("LYR").Cells(R1, AClm) + Ang    ' Angle
                T1 = Sheets("LYR").Cells(R1, TClm)          ' Thickness

                If A1 >= 180 Then    ' Angle adjustment
                    A1 = A1 - 180
                End If

                Sheets("Code").Cells(CRow, CClm) = _
                    "RMORE," & M1 & "," & A1 & "," & T1 & ",,,"
                CRow = CRow + 1
             Else
                M1 = Sheets("LYR").Cells(R1, MClm)          ' Material #
                T1 = Sheets("LYR").Cells(R1, TClm)          ' Thickness
                A1 = Sheets("LYR").Cells(R1, AClm) + Ang    ' Angle
                M2 = Sheets("LYR").Cells(R2, MClm)          ' Material #
                T2 = Sheets("LYR").Cells(R2, TClm)          ' Thickness
                A2 = Sheets("LYR").Cells(R2, AClm) + Ang    ' Angle

                If A1 >= 180 Then    ' Angle adjustment
                    A1 = A1 - 180
                End If

                If A2 >= 180 Then    ' Angle adjustment
                    A2 = A2 - 180
                End If

                Sheets("Code").Cells(CRow, CClm) = _
                    "RMORE," & M1 & "," & A1 & "," & T1 & "," & M2 & "," & A2 & "," & T2
                CRow = CRow + 1
            End If
        Next

        ' Setting up for End Material
        Sheets("Code").Cells(CRow, CClm) = "R,2,1,,,,"
```

```
                CRow = CRow + 1
                Sheets("Code").Cells(CRow, CClm) = "RMORE,,,,,,"
                CRow = CRow + 1
                Sheets("Code").Cells(CRow, CClm) = "RMORE,2,0," & TEnd & ",,,"
                CRow = CRow + 1

            ElseIf ETYP = "SHELL91" Then ' If Shell91 elemet is used
                Sheets("Code").Cells(CRow, CClm) = "R,1," & NumLYR & ",1,,,"
                CRow = CRow + 1
                Sheets("Code").Cells(CRow, CClm) = "RMORE,,,,,,"
                CRow = CRow + 1

                For LYR = 1 To NumLYR

                    R1 = RowLast - LYR + 1

                    M1 = Sheets("LYR").Cells(R1, MClm)      ' Material #
                    T1 = Sheets("LYR").Cells(R1, TClm)      ' Thickness
                    A1 = Sheets("LYR").Cells(R1, AClm)      ' Angle

                    Sheets("Code").Cells(CRow, CClm) = _
                        "RMORE," & M1 & "," & A1 & "," & T1 & "," & T1 & "," & T1 & "," & T1
                    CRow = CRow + 1
                Next

                ' Setting up for End Material
                Sheets("Code").Cells(CRow, CClm) = "R,2,1,,,,"
                CRow = CRow + 1
                Sheets("Code").Cells(CRow, CClm) = "RMORE,,,,,,"
                CRow = CRow + 1
                Sheets("Code").Cells(CRow, CClm) = _
                        "RMORE,2,0," & TEnd & "," & TEnd & "," & TEnd & "," & TEnd
                CRow = CRow + 1
            End If

    ' Generates the Major Nodes -------------------------------------------------------------
        For Xdir = 1 To NEX + 1
                X = Round(X0 + dx * (Xdir - 1), 16)
            For YDir = 1 To NEY + 1
                Y = Round(Y0 + dy * (YDir - 1), 16)
                Sheets("Code").Cells(CRow, CClm) = "N,," & X & "," & Y & "," & 0
                CRow = CRow + 1
            Next
        Next
    ' Generates the Minor Nodes -------------------------------------------------------
        For Xdir = 1 To NEX + 1
                X = Round(X0 + dx * (Xdir - 1), 16)
            For YDir = 1 To NEY
                Y = Round(Y0 + dy * (YDir - 1) + dy / 2, 16)
                Sheets("Code").Cells(CRow, CClm) = "N,," & X & "," & Y & "," & 0
                CRow = CRow + 1
            Next
        Next
    ' Generates the Midpoint Nodes ---------------------------------------------------------
        For Xdir = 1 To NEX
                X = Round(X0 + dx * (Xdir - 1) + dx / 2, 16)
            For YDir = 1 To NEY + 1
                Y = Round(Y0 + dy * (YDir - 1), 16)
                Sheets("Code").Cells(CRow, CClm) = "N,," & X & "," & Y & "," & 0
                CRow = CRow + 1
            Next
        Next
    ' Generates the Elements ---------------------------------------------------------------
        For Xdir = 1 To NEX

            ' Choosing the real constant Set
            If NEXE = 0 Then ' No Ends Elements
                If Xdir = 1 Then
                    Sheets("Code").Cells(CRow, CClm) = "REAL,1"
                    CRow = CRow + 1
                End If
            ElseIf NEXE <> 0 Then ' Ends Elements
                If Xdir = 1 Then
                    Sheets("Code").Cells(CRow, CClm) = "REAL,2"
                    CRow = CRow + 1
                ElseIf Xdir = 1 + NEXE Then
                    Sheets("Code").Cells(CRow, CClm) = "REAL,1"
                    CRow = CRow + 1
                ElseIf Xdir = NEX - (NEXE - 1) Then
                    Sheets("Code").Cells(CRow, CClm) = "REAL,2"
                    CRow = CRow + 1
```

```
                End If
            End If

            For YDir = 1 To NEY

                Ni = (Xdir - 1) * (NEY + 1) + YDir  ' Node Number of "i"

                I = Ni                              ' Node I
                J = Ni + (NEY + 1)                  ' Node J
                K = Ni + (NEY + 1) + 1              ' Node K
                L = Ni + 1                          ' Node L
                M = Ni + (NMJR + NMNR)              ' Node M
                N = Ni + NMJR + NEY - (Xdir - 1)    ' Node N
                O = Ni + (NMJR + NMNR) + 1          ' Node O
                P = Ni + NMJR - (Xdir - 1)          ' Node P

                'Code Output
                Sheets("Code").Cells(CRow, CClm) = _
                "E," & I & "," & J & "," & K & "," & L & "," & M & "," & N & "," & O & "," & P
                CRow = CRow + 1
            Next
        Next

' Apply Constraints -----------------------------------------------------
    ' Plane Stress mode
        If PSS = "Y" Then ' Plane Stress Switch
            Sheets("Code").Cells(CRow, CClm) = "NSEL,ALL"
            CRow = CRow + 1
            Sheets("Code").Cells(CRow, CClm) = "D,ALL" & ",UZ,0,,,,ROTX,ROTY,,,"
            CRow = CRow + 1
        End If
    ' Constrains in X direction at root ------------------------------
        For N = 1 To NEY + 1 ' Major nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UX,0,,,,,,,,"
            CRow = CRow + 1
        Next

        For N = 1 + NMJR To NMJR + NEY ' Ninor nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UX,0,,,,,,,,"
            CRow = CRow + 1
        Next
    ' Apply Coupling at tip -----------------------------------------
        NBase = 1 + (NEY + 1) * NEX
        ' Major Nodes
        For N = 1 + (NEY + 1) * NEX + 1 To 1 + (NEY + 1) * NEX + NEY
            Sheets("Code").Cells(CRow, CClm) = "CP,1,UX," & NBase & "," & N
            CRow = CRow + 1
        Next

        ' Minor Nodes
        For N = NMJR + NMNR - NEY + 1 To NMJR + NMNR
            Sheets("Code").Cells(CRow, CClm) = "CP,1,UX," & NBase & "," & N
            CRow = CRow + 1
        Next

    ' End Grips for UY and UZ

    If GTYP = 1 Then '----------------------------------------------------------------
        ' This boundary condition has UY=0 and UZ=0 at all place at both ends.
        ' The UX=0 at X=0

        ' Root
        For N = 1 To NEY + 1 ' Major nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,UY,ROTX,ROTY,,"
            CRow = CRow + 1
        Next

        For N = 1 + NMJR To NMJR + NEY ' Ninor nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,UY,ROTX,ROTY,,"
            CRow = CRow + 1
        Next

        ' Tip
        For N = 1 + (NEY + 1) * (NEX) To 1 + (NEY + 1) * NEX + NEY ' Major nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,UY,ROTX,ROTY,,"
            CRow = CRow + 1
        Next

        For N = 1 + NMJR + NMNR - NEY To NMJR + NMNR ' Minor nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,UY,ROTX,ROTY,,"
            CRow = CRow + 1
```

```
        Next

ElseIf GTYP = 2 Then '----------------------------------------------------------------
    ' This boundary condition has UZ=0 at all place at both ends
    ' The UX=0 at X=0 The UY will be set to 0 at middle of the coupon at the ends

    ' Root
    For N = 1 To NEY + 1 ' Major nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    For N = 1 + NMJR To NMJR + NEY ' Ninor nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    ' Tip
    For N = 1 + (NEY + 1) * (NEX) To 1 + (NEY + 1) * NEX + NEY ' Major nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    For N = 1 + NMJR + NMNR - NEY To NMJR + NMNR ' Minor nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

        NCR = 1 + NEY / 2                        ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UY,0,,,,UZ,,,,"
        CRow = CRow + 1

        NCT = 1 + NEY / 2 + (NEY + 1) * (NEX) ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UY,0,,,,UZ,,,,"
        CRow = CRow + 1

ElseIf GTYP = 3 Then '----------------------------------------------------------------
    ' This boundary condition has UZ=0 at all place at both ends
    ' The UX=0 at X=0 The UY will be set to 0 at middle of X=0

    ' Root
    For N = 1 To NEY + 1 ' Major nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    For N = 1 + NMJR To NMJR + NEY ' Ninor nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

        NCR = 1 + NEY / 2                        ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UY,0,,,,UZ,,,,"
        CRow = CRow + 1

    ' Tip
    For N = 1 + (NEY + 1) * (NEX) To 1 + (NEY + 1) * NEX + NEY ' Major nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    For N = 1 + NMJR + NMNR - NEY To NMJR + NMNR    ' Minor nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

ElseIf GTYP = 4 Then '----------------------------------------------------------------
    ' This boundary condition has UZ=0 at middle of both ends
    ' The UX=0 at X=0 The UY will be set to 0 at middle of the coupon at the ends

        NCR = 1 + NEY / 2                        ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UZ,0,,,,,,,,"
        CRow = CRow + 1

        NCT = 1 + NEY / 2 + (NEY + 1) * NEX        ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UZ,0,,,,,,,,"
        CRow = CRow + 1

        NCR = 1 + NEY / 2                        ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UY,0,,,,UZ,,,,"
        CRow = CRow + 1
```

```vba
        NCT = 1 + NEY / 2 + (NEY + 1) * (NEX)         ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1

ElseIf GTYP = 5 Then '-----------------------------------------------------------------
    ' This boundary condition has UZ=0 at middle of both ends
    ' The UY will be set to 0 at middle of the coupon at the X=0
    ' The UX=0 at X=0 The UY will be set to 0 at middle of the coupon at the ends

        NCR = 1 + NEY / 2                             ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UZ,0,,,,,,,,"
        CRow = CRow + 1

        NCT = 1 + NEY / 2 + (NEY + 1) * NEX           ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UZ,0,,,,,,,,"
        CRow = CRow + 1

        NCR = 1 + NEY / 2                             ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1

        NCT = 1 + NEY / 2 + (NEY + 1) * (NEX)         ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1

ElseIf GTYP = 6 Then '-----------------------------------------------------------------
    ' This boundary condition has UZ=0 at middle of both ends
    ' The UY will be set to 0 at middle of the coupon at the X=0
    ' The UX=0 at X=0 The UY will be set to 0 at middle of the coupon at the ends

    ' Root
    For N = 1 To NEY + 1 ' Major nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    For N = 1 + NMJR To NMJR + NEY ' Ninor nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

        NCT = 1 + NEY / 2 + (NEY + 1) * NEX      ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UZ,0,,,,,,,,"
        CRow = CRow + 1

        NCR = 1 + NEY / 2                        ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1

        NCT = 1 + NEY / 2 + (NEY + 1) * (NEX)    ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1

ElseIf GTYP = 7 Then '-----------------------------------------------------------------
    ' This boundary condition has UZ=0 at middle of both ends
    ' The UY will be set to 0 at middle of the coupon at the X=0
    ' The UX=0 at X=0 The UY will be set to 0 at middle of the coupon at the ends

    ' Root
    For N = 1 To NEY + 1 ' Major nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

    For N = 1 + NMJR To NMJR + NEY ' Ninor nodes
        Sheets("Code").Cells(CRow, CClm) = "D," & N & ",UZ,0,,,,,ROTX,ROTY,,,"
        CRow = CRow + 1
    Next

        NCT = 1 + NEY / 2 + (NEY + 1) * NEX      ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UZ,0,,,,,,,,"
        CRow = CRow + 1

        NCR = 1 + NEY / 2                        ' Node # at Center at Root
        Sheets("Code").Cells(CRow, CClm) = "D," & NCR & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1

        NCT = 1 + NEY / 2 + (NEY + 1) * (NEX)    ' Node # at Center at Tip
        Sheets("Code").Cells(CRow, CClm) = "D," & NCT & ",UY,0,,,,,UZ,,,,,"
        CRow = CRow + 1
```

```
        For N = 1 + (NEY + 1) * (NEX) To 1 + (NEY + 1) * NEX + NEY ' Major nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",ROTY,0,,,,,,,,"
            CRow = CRow + 1
        Next

        For N = 1 + NMJR + NMNR - NEY To NMJR + NMNR     ' Minor nodes
            Sheets("Code").Cells(CRow, CClm) = "D," & N & ",ROTY,0,,,,,,,,"
            CRow = CRow + 1
        Next

    End If

' Delete all loadings ------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "NSEL,ALL"    ' Select all nodes
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "FDELE,ALL"   ' delete all forces
    CRow = CRow + 1
    NCT = 1 + NEY / 2 + (NEY + 1) * NEX    ' Node # at Center at Tip
    Sheets("Code").Cells(CRow, CClm) = "F," & NCT & ",FX," & LD
    CRow = CRow + 1

' Solve The Model ----------------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "NSEL,ALL"    ' Select all nodes
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "/SOLU"       ' mode to solution mode
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "ANTYPE,STATIC,NEW"  ' static analysis
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "PSTRES,ON"   ' Turn on prestress
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "SOLVE"       ' Solve the model
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "FINISH"      ' Exit from solution mode
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "/POST1"      ' Read the result
    CRow = CRow + 1

' Gerring the pictures out ------------------------------------
    Sheets("Code").Cells(CRow, CClm) = "NSEL,ALL"    ' Select all nodes
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "ESEL,ALL"    ' Select all nodes
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "JPEG,COLOR,2"    ' Output in color
    CRow = CRow + 1
    Sheets("Code").Cells(CRow, CClm) = "/SHOW,JPEG"      ' Open JPEG mode
    CRow = CRow + 1

    For PicCase = 1 To 17

        If PicCase = 1 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,U,X,2"     ' Displacement X
        ElseIf PicCase = 2 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,U,Y,2"     ' Displacement Y
        ElseIf PicCase = 3 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,U,Z,2"     ' Displacement Y
        ElseIf PicCase = 4 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,ROT,X,2"   ' Displacement X
        ElseIf PicCase = 5 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,ROT,Y,2"   ' Displacement Y
        ElseIf PicCase = 6 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,S,X,2"     ' Stresses X
        ElseIf PicCase = 7 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,S,Y,2"     ' Stresses Y
        ElseIf PicCase = 8 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,S,Z,2"     ' Stresses Y
        ElseIf PicCase = 9 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,S,XY,2"    ' Stresses XY
        ElseIf PicCase = 10 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,S,YZ,2"    ' Stresses XY
        ElseIf PicCase = 11 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,S,XZ,2"    ' Stresses XY
        ElseIf PicCase = 12 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,EPEL,X,2" ' Strains X
        ElseIf PicCase = 13 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,EPEL,Y,2" ' Strains Y
        ElseIf PicCase = 14 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,EPEL,Z,2" ' Strains Y
        ElseIf PicCase = 15 Then
            Sheets("Code").Cells(CRow, CClm) = "PLNSOL,EPEL,XY,2" ' Strains XY
        ElseIf PicCase = 16 Then
```

```
                    Sheets("Code").Cells(CRow, CClm) = "PLNSOL,EPEL,YZ,2"   ' Strains XY
            ElseIf PicCase = 17 Then
                    Sheets("Code").Cells(CRow, CClm) = "PLNSOL,EPEL,XZ,2"   ' Strains XY
            End If
                    CRow = CRow + 1
                    Sheets("Code").Cells(CRow, CClm) = "/SHOW,CLOSE"    ' Close JPEG mode
                    CRow = CRow + 1
         Next

    ' Getting the result out -------------------------------------------
        ' Stresses
        Sheets("Code").Cells(CRow, CClm) = "NSEL,ALL"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "/OUTPUT," & FNAME & ",S"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "PRNSOL,S"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "/OUTPUT"
        CRow = CRow + 1
        ' Strains
        Sheets("Code").Cells(CRow, CClm) = "/OUTPUT," & FNAME & ",E"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "PRNSOL,EPEL"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "/OUTPUT"
        CRow = CRow + 1
        ' DOF
        Sheets("Code").Cells(CRow, CClm) = "/OUTPUT," & FNAME & ",D"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "PRNSOL,DOF"
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "/OUTPUT"
        CRow = CRow + 1

    ' Ending the session -----------------------------------------------
        Sheets("Code").Cells(CRow, CClm) = "FINISH" ' out of solution mode
        CRow = CRow + 1
        Sheets("Code").Cells(CRow, CClm) = "SAVE"    ' Save the model
        CRow = CRow + 1

    End If
Next

Application.ScreenUpdating = True ' Turn back on screen update

Sheets("Code").Cells(1, 1).Select
MsgBox ("Done!")

End Sub
```

Postprocessing Macro

The numerical result data can be acquired from the solution. This macro code will reorganize the raw file into a spreadsheet.

```
Sub RetRawData()
' This macro is to retrieve the raw data and reorganize into
' a format of uers's wish

Application.ScreenUpdating = False

' Title
    Cells(1, 1) = Cells(9, 1)

    Row1 = 0    ' Raw Data Row #
    Row2 = 0    ' Block Raw Data Row #
    Row3 = 7    ' Summary Row #
    Row4 = 0    ' Retrieving Row #

' Determine the Stresses of Strains

If Mid(Cells(2, 1), 8, 1) = "E" Then
    ' Label the row (Strains)
    Cells(6, 1) = "Node"
    Cells(6, 2) = "Ex"
    Cells(6, 3) = "Ey"
```

L

```
        Cells(6, 4) = "Ez"
        Cells(6, 5) = "Exy"
        Cells(6, 6) = "Eyz"
        Cells(6, 7) = "Exz"
        Cells(5, 9) = "EPELX"
        Cells(5, 16) = "EPELY"
        Cells(5, 23) = "EPELZ"
        Cells(5, 30) = "EPELXY"
        Cells(5, 37) = "EPELYZ"
        Cells(5, 44) = "EPELXZ"
        Sufix = "_E"
        FRN = 26      ' First Row #
        STP = 59      ' Step
        PAK = 36      ' Packet

    ElseIf Mid(Cells(2, 1), 8, 1) = "S" Then
        ' Label the row (Stresses)
        Cells(6, 1) = "Node"
        Cells(6, 2) = "Sx"
        Cells(6, 3) = "Sy"
        Cells(6, 4) = "Sz"
        Cells(6, 5) = "Sxy"
        Cells(6, 6) = "Syz"
        Cells(6, 7) = "Sxz"
        Cells(5, 9) = "SX"
        Cells(5, 16) = "SY"
        Cells(5, 23) = "SZ"
        Cells(5, 30) = "SXY"
        Cells(5, 37) = "SYZ"
        Cells(5, 44) = "SXZ"
        Sufix = "_S"
        FRN = 26      ' First Row #
        STP = 59      ' Step
        PAK = 36      ' Packet

    ElseIf Mid(Cells(2, 1), 8, 1) = "D" Then
        ' Label the row (DOF)
        Cells(6, 1) = "Node"
        Cells(6, 2) = "UX"
        Cells(6, 3) = "UY"
        Cells(6, 4) = "UZ"
        Cells(6, 5) = "ROTX"
        Cells(6, 6) = "ROTY"
        Cells(6, 7) = "ROTZ"
        Cells(5, 9) = "UX"
        Cells(5, 16) = "UY"
        Cells(5, 23) = "UZ"
        Cells(5, 30) = "ROTX"
        Cells(5, 37) = "ROTY"
        Cells(5, 44) = "ROTZ"
        Sufix = "_D"
        FRN = 25      ' First Row #
        STP = 59      ' Step
        PAK = 37      ' Packet

    End If

        Cells(2, 1) = "L"    ' Length of the coupon
        Cells(3, 1) = "L/W" ' Clear the second row
        Cells(4, 1) = "# of Elem in X" ' Clear the second row
        Cells(5, 1) = "# of Elem in Y" ' Clear the second row

    ' Transferring the data

        For Row1 = FRN To 60000 Step STP

            For Row2 = 1 To PAK Step 1

                    Row4 = Row1 + Row2 - 1
                    ' When to Stop
                    If Mid(Cells(Row4, 1), 2, 7) = "MINIMUM" Then
                        Exit For
                    End If

                    Cells(Row3, 1) = Mid(Cells(Row4, 1), 1, 8)    ' Node#
                    Cells(Row3, 2) = Mid(Cells(Row4, 1), 10, 12) ' X
                    Cells(Row3, 3) = Mid(Cells(Row4, 1), 22, 12) ' Y
                    Cells(Row3, 4) = Mid(Cells(Row4, 1), 34, 12) ' Z
                    Cells(Row3, 5) = Mid(Cells(Row4, 1), 46, 12) ' XY
                    Cells(Row3, 6) = Mid(Cells(Row4, 1), 58, 12) ' YZ
                    Cells(Row3, 7) = Mid(Cells(Row4, 1), 70, 12) ' XZ
```

LI

```
            Row3 = Row3 + 1 ' Summary row number increased
        Next
    Next

    Columns("B:G").Select
    Selection.Columns.AutoFit

    A = InStr(Cells(1, 1), "_LW")   ' Find Aspect Ratio
    B = InStr(Cells(1, 1), "_G")    ' Find Grip Type
    C = InStr(Cells(1, 1), "_X")    ' Find X
    D = InStr(Cells(1, 1), "_Y")    ' Find Y
    E = InStrRev(Cells(1, 1), "_")  ' Find RC
    F = InStr(Cells(1, 1), "  ") - 2

    LW = Mid(Cells(1, 1), A + 3, B - (A + 3))
    L = LW
    NEX = Mid(Cells(1, 1), C + 2, D - (C + 2))
    NEY = Mid(Cells(1, 1), D + 2, E - (D + 2))

    Cells(2, 2) = L      ' "L"   ' Length of the coupon
    Cells(3, 2) = LW     ' "L/W" ' Clear the second row
    Cells(4, 2) = NEX    ' "# of Elem in X"
    Cells(5, 2) = NEY    ' "# of Elem in Y"

    dx = L / NEX
    dy = NEY / 4

For ClmRst = 9 To 45 Step 7

    Cells(6, ClmRst) = "X"          ' X
    Cells(6, ClmRst + 1) = "Y=-W/2" ' Location 1
    Cells(6, ClmRst + 2) = "Y=-W/4" ' Location 2
    Cells(6, ClmRst + 3) = "Y=0"    ' Location 3
    Cells(6, ClmRst + 4) = "Y=W/4"  ' Location 4
    Cells(6, ClmRst + 5) = "Y=W/2"  ' Location 5

    Row1 = 7    ' Starting Row For Result

    For Turns = 1 To NEX + 1
        X = dx * (Turns - 1)        ' X value
        Node1 = 1 + (NEY + 1) * (Turns - 1) ' First Node #
        Row2 = Node1 + 6            ' Location of the Row in the RawData
        ClmRst2 = (ClmRst - 2) / 7 + 1  ' Location of the column

        Cells(Row1, ClmRst) = X
        Cells(Row1, ClmRst + 1) = Cells(Row2, ClmRst2)
        Cells(Row1, ClmRst + 2) = Cells(Row2 + dy, ClmRst2)
        Cells(Row1, ClmRst + 3) = Cells(Row2 + dy * 2, ClmRst2)
        Cells(Row1, ClmRst + 4) = Cells(Row2 + dy * 3, ClmRst2)
        Cells(Row1, ClmRst + 5) = Cells(Row2 + dy * 4, ClmRst2)

        Row1 = Row1 + 1
    Next
Next

Application.ScreenUpdating = True

    Filename = Mid(Cells(1, 1), 2, F) & Sufix & ".xls"

    ActiveWorkbook.SaveAs Filename:=Filename, FileFormat:= _
        xlNormal, Password:="", WriteResPassword:="", ReadOnlyRecommended:=False _
        , CreateBackup:=False

ActiveWorkbook.Close

MsgBox ("Done")

End Sub
```

Appendix E: References

References:

1. "More on the Influence of End Constraints on Off-Axis Tensile Tests" written by R. R. Rizzo.
2. "Design and Testing of composite Laminates for Optimum Extension-Twist Coupling" written by Erian A. Armanious, David Hooke, Manahar Kamai, David Palmer and Jian Li.
3. Mechanics of Composite Materials, Robert. M. Jones, 1975, McGraw Hill Book Company, New York, ISBN 0-07-032790-4
4. ANSYS Help System
5. AE653 Basic Composite Material handouts
6. http://www.omega.com/literature/transactions/volume3/strain/html The Strain Gage
7. http://www.efunda.com/formulae/solid_mechanics/composites/comp_laminates.cfm Classical Lamination Theory

List of Material Properties:

1. Unidirectional Carbon   Reference: Sandia & GE's windturbine project
   - $EX = 1.20*10^{11}$ Pa
   - $EY = 9.33*10^{9}$ Pa
   - $GXY = 4.67*10^{9}$ Pa
   - $\nu12 = 0.280$
2. Unidirectional Boron   Reference: Reference #1 (pp.207)
   - $EX = 2.62*10^{11}$ Pa
   - $EY = 2.76*10^{10}$ Pa
   - $GXY = 6.90*10^{9}$ Pa
   - $\nu12 = 0.240$
3. AS4/3502 Prepreg   Reference : Reference #2 (Table 3, pp.259)
   - $EX = 1.31*10^{11}$ Pa
   - $EY = 1.06*10^{10}$ Pa
   - $GXY = 5.93*10^{9}$ Pa
   - $\nu12 = 0.33$
4. Unidirectional E-glass   Reference: Sandia & GE's windturbine project
   - $EX = 3.97*10^{10}$ Pa
   - $EY = 8.40*10^{9}$ Pa
   - $GXY = 4.30*10^{9}$ Pa
   - $\nu12 = 0.29$
5. Biax $\pm$45degrees   Reference: Sandia & GE's windturbine project
   - $EX = 9.18*10^{9}$ Pa
   - $EY = 9.18*10^{9}$ Pa
   - $GXY = 9,.91*10^{9}$ Pa
   - $\nu12 = 0.637$