

Composite Laminate Modeling

Handbook for FEMAP, NX Nastran and LS-DYNA Users

Adrian Jensen, PE, Senior Staff Mechanical Engineer

George Laird, PhD, PE, Principal Mechanical Engineer



**LS-DYNA Sales, Support
& Consulting**

www.PredictiveEngineering.com



Siemens PLM Software Sales & Support
CAD | CAM | CAE | Teamcenter

www.AppliedCAx.com

WHAT THIS HANDBOOK COVERS

This note is intended for new engineers interested in modeling composites and experienced engineers who would like to get acquainted with the FEMAP interface.

The following topics are covered:

- A little background on the mechanics of composites and how micromechanics can be leveraged to obtain composite material properties
- 2D composite laminate modeling
 - Defining a material model, layup, property card and material angles
 - Symmetric vs. unsymmetric laminate and why this is important
 - Results post processing
- 3D composite laminate modeling
 - Defining a material model, layup, property card and ply/stack orientation
 - When is a 3D model preferred over a 2D model
- Modeling a sandwich composite
 - Methods of modeling a sandwich composite
 - 3D vs. 2D sandwich composite models and their pros and cons
- Failure modeling of a 2D composite laminate
 - Defining a laminate failure model
 - Post processing laminate and lamina failure indices
- Additional examples and theory

TABLE OF CONTENTS

1.	INTRODUCTION.....	7
1.1	TERMINOLOGY	7
1.2	TYPES OF MATERIALS	8
1.2.1	Anisotropic Material	8
1.2.2	Orthotropic Material.....	9
1.2.3	Isotropic Material	9
2.	COMPOSITE MICROMECHANICS	10
2.1	RULE OF MIXTURES	10
2.1.1	Application Example	10
3.	LAMINATE MODELING IN FEMAP	12
3.1	IMPORTANT ENTITIES	13
3.2	DEFINING AN ORTHOTROPIC MATERIAL	14
3.3	UNDERSTANDING THE LAYUP EDITOR	15
3.4	PROPERTY CARD: 2D LAMINATE MODELING.....	17
3.5	ASSIGNING MATERIAL ANGLES.....	18
3.5.1	Defining a Material Angle on the Element	19
3.5.2	Defining a Material Angle in the Property Card	20
3.5.3	Advanced Application Example of Assigning Material Angle to a Complex Geometry.....	21
3.6	PROPERTY CARD: 3D LAMINATE MODELING.....	25
3.7	MULTI-MATERIAL ANGLE EXAMPLE OF A 3D COMPOSITE MODEL	26
4.	EXAMPLE 1: CREATING A 2D LAMINATE MODEL IN FEMAP	29
4.1	INTRODUCTION	29

4.2	CREATING THE MATERIAL PROPERTY	29
4.3	DEFINING THE LAYUP	30
4.4	DEFINING THE LAMINATE PROPERTY (PCOMP)	31
4.5	SPECIFYING MATERIAL ANGLES	32
4.6	ANALYSIS SETUP AND POST PROCESSING.....	32
4.6.1	Analysis Set Manager Setup for Composites.....	34
4.6.2	Post Processing the Results	35
5.	EXAMPLE 2: CREATING A 3D LAMINATE MODEL IN FEMAP	39
5.1	INTRODUCTION	39
5.2	CREATING THE MATERIAL PROPERTY	40
5.3	DEFINING THE LAYUP	40
5.4	DEFINING THE LAMINATE PROPERTY (PCOMPS)	40
5.5	SPECIFYING MATERIAL ANGLES	41
5.6	POST PROCESSING THE RESULTS	41
6.	EXAMPLE 3: MODELING A SANDWICH COMPOSITE	43
6.1	INTRODUCTION	43
6.2	CREATING THE MATERIAL PROPERTY	43
6.3	DEFINING THE LAYUP	43
6.4	DEFINING THE PROPERTY CARDS	43
6.5	POST PROCESSING THE RESULTS	44
6.6	OTHER METHODS FOR SANDWICH COMPOSITE MODELING	46
7.	LAMINATE FAILURE THEORIES IN FEMAP	48
7.1	INTRODUCTION	48
7.2	HILL'S THEORY.....	48

7.3	HOFFMAN’S THEORY	49
7.4	TSAI-WU THEORY	50
7.5	MAXIMUM STRAIN THEORY.....	50
7.6	ONSET FAILURE THEORY	51
8.	EXAMPLE 4: MODELING THE FAILURE BEHAVIOR OF COMPOSITE LAMINATES.....	51
8.1	CREATING THE MATERIAL PROPERTY	51
8.2	DEFINING THE LAMINATE PROPERTY	52
8.3	RESULTS	52
9.	ADDITIONAL READING	57
10.	MATERIAL DATABASE.....	58
11.	FOUR-POINT BENDING OF A SANDWICH COMPOSITE USING FEMAP AND NX NASTRAN.....	61
11.1	HAND CALCULATION	62
11.1.1	Basic Laminate	62
11.1.2	Sandwich Composite.....	71
11.2	FINITE ELEMENT SIMULATION	77
11.3	SUMMARY OF RESULTS	86
12.	ADDENDUM.....	88
12.1	EXAMPLE 1 (2D LAMINATE WITH A HOLE).....	88
12.1.1	Displacement Contour	88
12.1.2	Major Principal Stress in Ply 4 (0°).....	89
12.1.3	Unsymmetric Laminate.....	90
12.2	EXAMPLE 2 (3D LAMINATE WITH A HOLE).....	91
12.2.1	Displacement Contour	91
12.2.2	Major Principal Stress in Ply 4 (0°).....	92

12.2.3	3D Solid Laminate with One Element for Each Layer Through Thickness	93
12.3	EXAMPLE 3 (3D SANDWICH COMPOSITE).....	95
12.3.1	Displacement Contour	95
12.3.2	Transverse Shear Stress (ZX) in Core	96
12.4	EXAMPLE 4 (2D LAMINATE WITH FAILURE).....	97
12.4.1	Laminate Failure	97
12.4.2	Failure in Longitudinal Tension.....	98
12.4.3	Failure in Transverse Tension	99
13.	APPENDIX.....	100
13.1	CLASSICAL LAMINATION THEORY.....	100
13.1.1	Kinematic Equations	100
13.1.2	Constitutive Equations.....	100
13.1.3	Resultants	101
13.1.4	Equilibrium Equations.....	101
13.1.5	Stiffness Matrices A, B, and D.....	101
13.2	UNSYMMETRIC LAYUP	103
13.3	CHAMIS MODEL	105
13.4	ONSET FAILURE THEORY	105

Composite Laminate Modeling Using FEMAP

1. INTRODUCTION

1.1 TERMINOLOGY

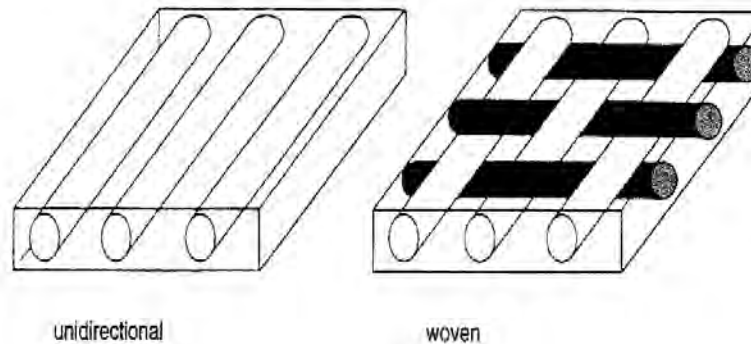
- Composite material: A combination of two or more materials to form a new material system with enhanced material properties.



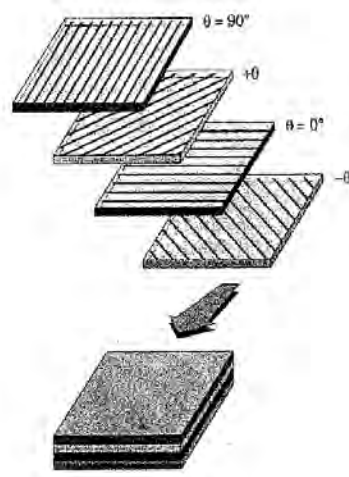
Examples of reinforcements are glass fibers, carbon fibers, silicon carbide fibers etc.

Examples of matrix materials are epoxy, polyurethane, silicon carbide etc.

- Lamina: A lamina is a thin layer of composite material. The thickness of the lamina is usually 0.1 to 1 mm. It is also referred to as a ply.



- Laminate: A laminate is constructed by stacking a number of laminae. The building block for a laminate is lamina.



1.2 TYPES OF MATERIALS

In engineering applications, materials can be broadly classified as anisotropic, orthotropic, and isotropic. An anisotropic material has the generalized formulation, while other two are derived by some simplifications.

1.2.1 ANISOTROPIC MATERIAL

An anisotropic material has no planes of material symmetry. Examples of anisotropic materials are femur, short-fiber composites etc. The number of constants required to describe anisotropic materials is 21. The stiffness matrix shown below is symmetric about the diagonal terms. Accordingly, all the diagonal terms and the terms above/below the diagonal terms have to be defined in the material model.

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{23} \\ \tau_{13} \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{12} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{13} & C_{23} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{14} & C_{24} & C_{34} & C_{44} & C_{45} & C_{46} \\ C_{15} & C_{25} & C_{35} & C_{45} & C_{55} & C_{56} \\ C_{16} & C_{26} & C_{36} & C_{46} & C_{56} & C_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \epsilon_3 \\ \gamma_{23} \\ \gamma_{13} \\ \gamma_{12} \end{Bmatrix}$$

1.2.2 ORTHOTROPIC MATERIAL

An orthotropic material has three planes of material symmetry. Examples of orthotropic materials are wood, unidirectional lamina etc. The number of constants required to describe orthotropic materials is 9. The stiffness matrix is symmetric and all the diagonal terms and terms above/below the diagonal terms have to be defined. However, commercial finite element software allow defining the elastic moduli (E_1, E_2, E_3), shear moduli (G_{12}, G_{13}, G_{23}), and Poisson's ratio ($\nu_{12}, \nu_{13}, \nu_{23}$) instead of calculating each of the stiffness terms. The finite element software can internally evaluate these stiffness terms.

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{23} \\ \tau_{13} \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \epsilon_3 \\ \gamma_{23} \\ \gamma_{13} \\ \gamma_{12} \end{Bmatrix}$$

1.2.3 ISOTROPIC MATERIAL

All planes are planes of symmetry. Examples of isotropic materials are metals like steel, aluminum etc. There are only 2 independent constants (C_{11} and C_{12}) for an isotropic material. Similar to the orthotropic material model, one can define E and ν instead of calculating the stiffness terms.

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{23} \\ \tau_{13} \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{11} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{12} & C_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{C_{11}-C_{12}}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{C_{11}-C_{12}}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{C_{11}-C_{12}}{2} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \epsilon_3 \\ \gamma_{23} \\ \gamma_{13} \\ \gamma_{12} \end{Bmatrix}$$

2. COMPOSITE MICROMECHANICS

- Study of composite material behavior wherein the interaction of constituent materials are examined on a microscopic scale to determine their effect on the properties of the composite material.
- Predict the properties of composite, given the component properties and their geometric arrangement.
- The mechanical properties of the composite depend on the percentages of fibers and matrix.

2.1 RULE OF MIXTURES

- If the fiber and matrix properties are available, a reasonable estimation of the lamina properties can be obtained using the rule of mixtures (strength of materials approach).

$$E_L = E_f V_f + E_m V_m$$

$$\frac{1}{E_T} = \frac{V_f}{E_f} + \frac{V_m}{E_m}$$

$$\frac{1}{G_{LT}} = \frac{V_f}{G_f} + \frac{V_m}{G_m}$$

$$\nu_{LT} = \nu_f V_f + \nu_m V_m$$

Here, E_L and E_T correspond to longitudinal and transverse moduli of the composite lamina, G_{LT} and ν_{LT} correspond to inplane shear modulus and Poisson's ratio, respectively. Suffix 'f' corresponds to fiber property and 'm' corresponds to matrix property. In the above equations, 'V' corresponds to volume fraction.

2.1.1 APPLICATION EXAMPLE

We will consider the mechanical properties of a carbon fiber and an epoxy resin to apply the rule of mixtures and estimate the lamina mechanical properties. For this example, we will consider a fiber volume fraction of 56%. For most structural applications, a fiber volume fraction greater than 55% is typically used. Increasing the fiber volume fraction can favor in terms of the load carrying capacity as fibers take majority of the load. However, fiber volume fraction cannot be increased beyond a certain limit (typically around 65%). Just as a note, the theoretical maximum volume fraction for a fiber (cylinder) is 78% for square packing. As the resin content is reduced, fibers are not

completely wet during the manufacturing process and can result in increased dry spots (part quality is reduced). This will result in reduced load transfer between the fibers.

Constituent	E_L (GPa)	E_T (GPa)	ν	G (GPa)	$V_f(\%)$
Carbon fiber	220	22	0.15	25	56
Epoxy resin	3.3	3.3	0.37	1.2	-

The carbon/epoxy lamina properties are calculated using the rule of mixtures as shown below.

$$E_L = 220 \cdot 0.56 + 3.3 \cdot 0.44 = \mathbf{125 \text{ GPa}}$$

$$1/E_T = (0.56/220) + (0.44/3.3) = 0.14 \text{ GPa}^{-1} \rightarrow E_T = \mathbf{7.4 \text{ GPa}}$$

$$1/G_{LT} = (0.56/25) + (0.44/1.2) = 0.39 \text{ GPa}^{-1} \rightarrow G_{LT} = \mathbf{2.6 \text{ GPa}}$$

$$\nu_{LT} = 0.56 \cdot 0.15 + 0.44 \cdot 0.37 = \mathbf{0.25}$$

The mechanical properties (2D orthotropic) obtained using the rule of mixtures for a carbon/epoxy lamina are shown below.

Carbon/epoxy lamina	E_L (GPa)	E_T (GPa)	G_{LT} (GPa)	ν_{LT}
Rule of mixtures	125	7.4	2.6	0.25
Chamis model	125	9.1	4.2	0.25
Experimental [1]	125	9.1	5	0.34

3. LAMINATE MODELING IN FEMAP

In this section, we provide details on modeling composite laminates in FEMAP. There are four important entities in which modeling laminates differs from conventional metallic components. Composite laminate modeling deals with the following modifications:

- an orthotropic/anisotropic (typically) material configuration
- laminates have a layered configuration, so a layup configuration has to be specified with the thickness of each layer and their orientation angles w.r.t a reference axis
- an element/property type that accounts for the layered configuration has to be used
- reference axis (material angle) specification

Each of the above entities and their definition in FEMAP is summarized in the following sub-sections. There are two work flows that can be followed to model a laminate in FEMAP depending on whether the laminate model is a 2D or 3D. Defining the material cards, layup, and property cards is common between 2D and 3D models; however, material angles can be specified using two approaches in a 2D model while only one approach is used for a 3D model. In 2D models material angles can be specified at element level or directly in the property cards. Specifying material angles on elements is a convenient approach especially for complex geometries; we can select a set of elements and apply the material angle instead of assigning a specific material angle to all elements in a property card. For 3D models, we have to define material angles in the property card and so we need to create multiple property cards for complex geometries. In the following sub-sections, an overview of FEMAP interface for laminate modeling is provided accompanied by worked examples.

Work flow 1	Work flow 2
laminate (2D)	laminate (2D) and solid laminate (3D)
material model	material model
layup	layup
property card	property card with material angle
define material angle on element	

3.1 IMPORTANT ENTITIES

Modeling composites can be easier if the following entities are carefully assigned.

It is important to select/create a coordinate system that is appropriate for your composite laminate. In most cases, the basic rectangular coordinate system will work. However, one needs to be careful while dealing with curved laminates. Individual ply orientations assigned in layup editor will be referenced/relative to coordinate system assigned in this step.

A material type for the composite laminate should be assigned. Typically used material types are 2D/3D orthotropic materials for continuous fiber-reinforced composite laminates.

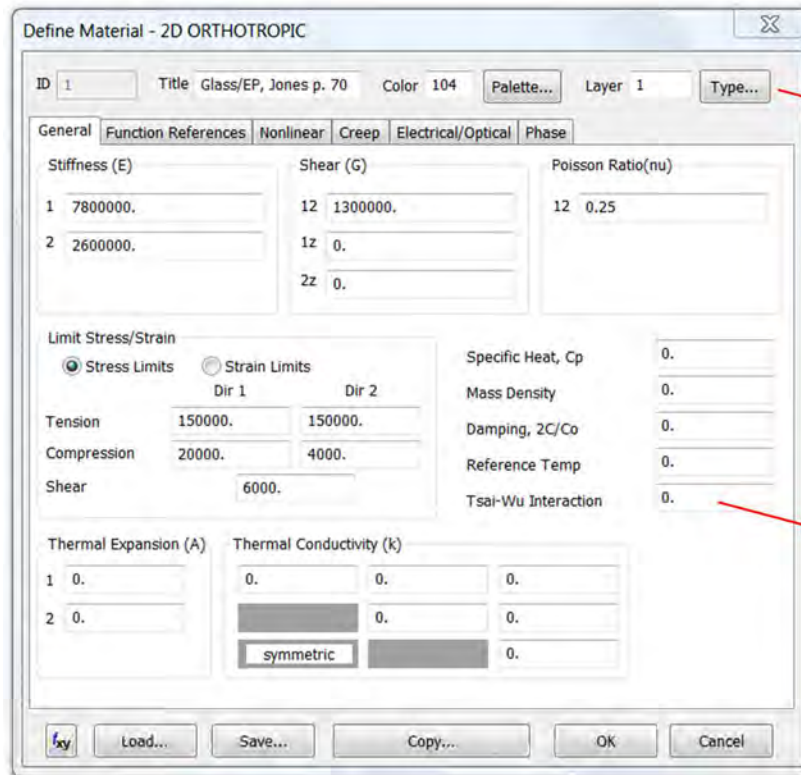
A property card corresponding to a composite laminate has to be specified. For 2D modeling, the element/property type can be a 'laminate' type. Similarly, for 3D modeling, the element/property type can be a 'solid laminate' type. A suitable failure criterion can also be specified using the property card.

The composite laminate layup configuration is provided using the layup editor. Using the layup editor, ply stacking (number and direction), ply material, individual ply orientations and ply thicknesses are provided.

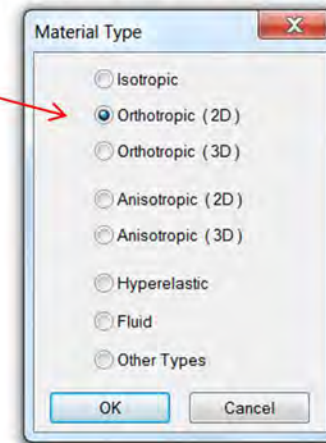
***Always check your units**

3.2 DEFINING AN ORTHOTROPIC MATERIAL

- Notice that we have different properties in longitudinal (E_1) and transverse (E_2) directions, where E_1 is aligned to the angle specified within the layup.
- Depending on the type of analysis, various material properties are required.



Select an appropriate material type (eg., orthotropic 2D)

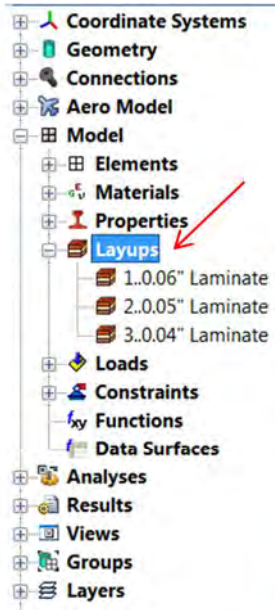


Required if Tsai-Wu failure criterion is used. This value is to be experimentally determined.

***Always check your units**

If you have fiber and matrix properties, you can still estimate the composite properties using rule of mixtures or other methods.

3.3 UNDERSTANDING THE LAYUP EDITOR



Thickness of individual ply

Global Ply ID (optional) AutoCreate Material Thickness Angle

0..None

--- Top of Layup, --- Total Thickness = 0.06

Ply ID	Global Ply	Material	Thickness	Angle
12		1..Glass/EP, Jones p. 70	0.005	45.
11		1..Glass/EP, Jones p. 70	0.005	-45.
10		1..Glass/EP, Jones p. 70	0.005	0.
9		1..Glass/EP, Jones p. 70	0.005	90.
8		1..Glass/EP, Jones p. 70	0.005	0.
7		1..Glass/EP, Jones p. 70	0.005	0.
6		1..Glass/EP, Jones p. 70	0.005	0.
5		1..Glass/EP, Jones p. 70	0.005	0.
4		1..Glass/EP, Jones p. 70	0.005	90.
3		1..Glass/EP, Jones p. 70	0.005	0.
2		1..Glass/EP, Jones p. 70	0.005	-45.
1		1..Glass/EP, Jones p. 70	0.005	45.

--- Bottom of Layup, ---

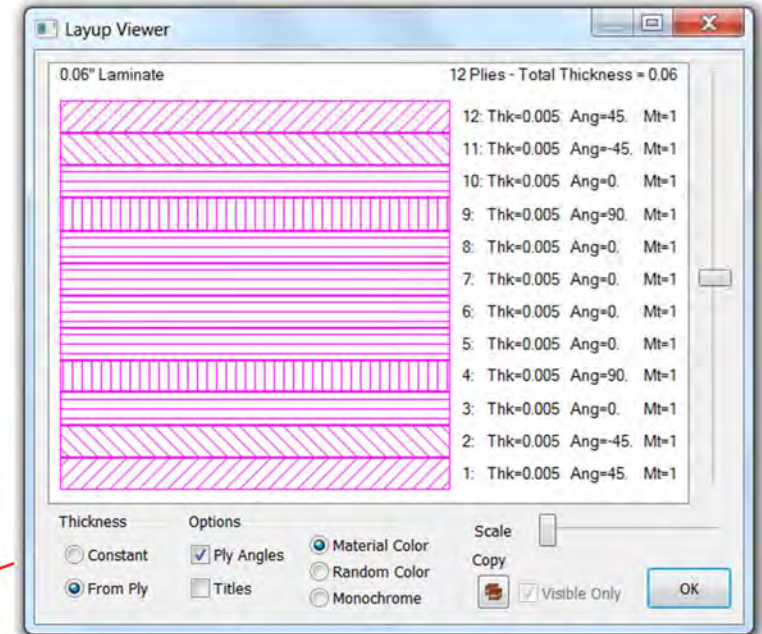
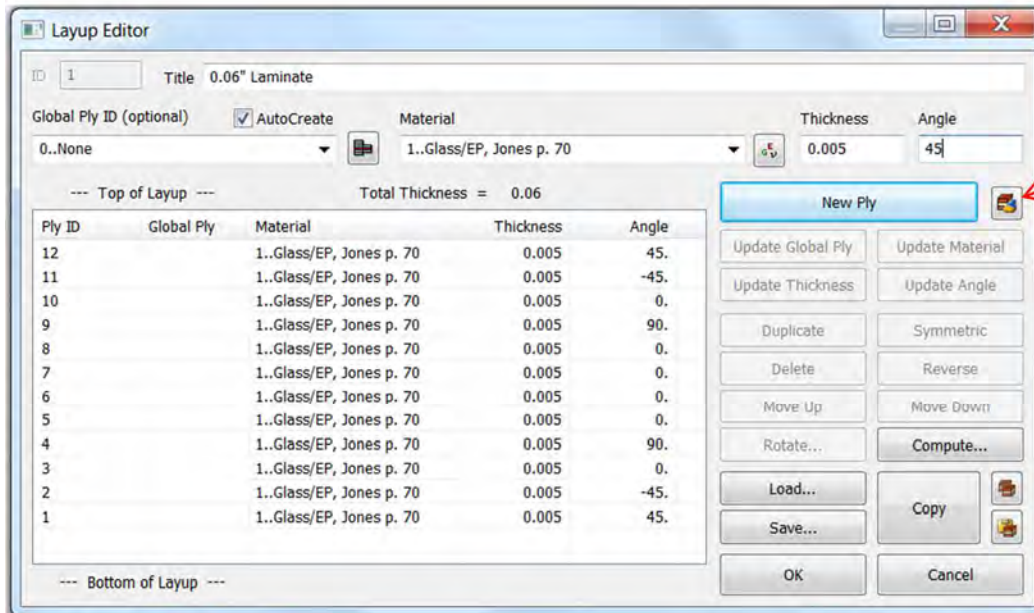
Notice the layup direction (numbering): bottom layer to top layer

Material assigned for each ply in the laminate. Multiple materials can also be assigned (eg., honeycomb core).

These angles/orientations are relative to the material angle (reference axis) specified for the element.

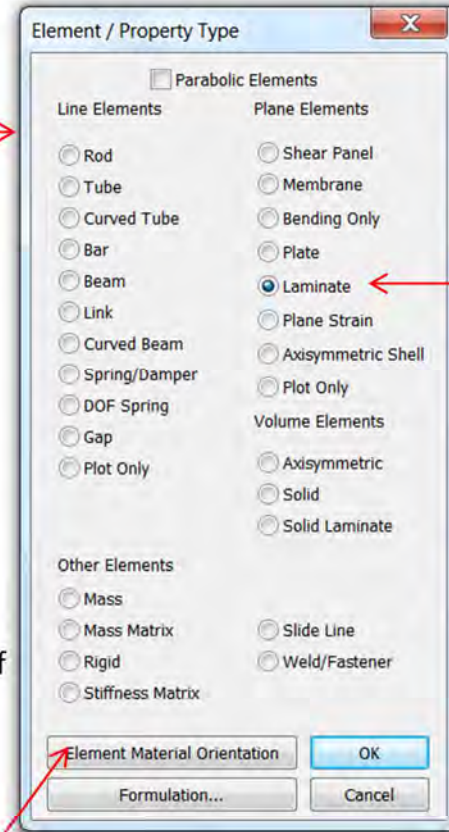
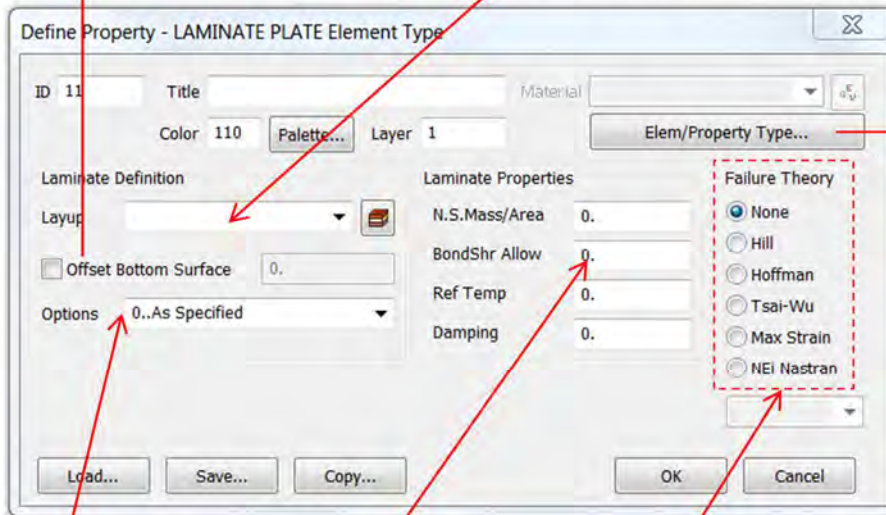
Computes laminate equivalent properties (inplane properties, bending properties, ABD matrices)

- Layup viewer is a useful tool to visualize the layup that has been created.
- It is important to notice that these angles/orientations are relative to the material angle (specified in the property card or on the elements). To avoid confusion, one can draw the orientations along with the assigned material angle and verify if the angles match the actual orientations in your layup.



3.4 PROPERTY CARD: 2D LAMINATE MODELING

Default is $-0.5 \times \text{total thickness}$ Select the layup created



2D laminate modeling

Required if a failure criterion is selected. This allowable stress corresponds to interlaminar shear stress.

Use a specific failure theory, if you want to compute failure indices of individual plies

Explore these options. If all the plies in the laminate are specified in the layup, you can leave it to default. If only half the plies are specified, you can use symmetric option to create a symmetric laminate.

Assign an appropriate material angle. Layup angles will be oriented relative to this material angle.

3.5 ASSIGNING MATERIAL ANGLES

Although, orientation angles are specified for each lamina individually in the layup editor, FEMAP will not understand as to which reference direction these orientations (angles) correspond. So, we need to specify a material angle such that all orientations (specified in layup) take this vector as the reference. For example, if the material angle is defined in the global x-direction then all the orientations will use global x-direction as the reference. One should not be confused with the terminologies 'material angle' and 'orientation angle' in the layup editor. Material angle is a reference axis that we assign to the element and is independent of the shape of the element. Orientation angle is the orientation of the lamina (fiber direction). By default, FEMAP assigns no material angle. Use any of the following methods to specify the material angle.

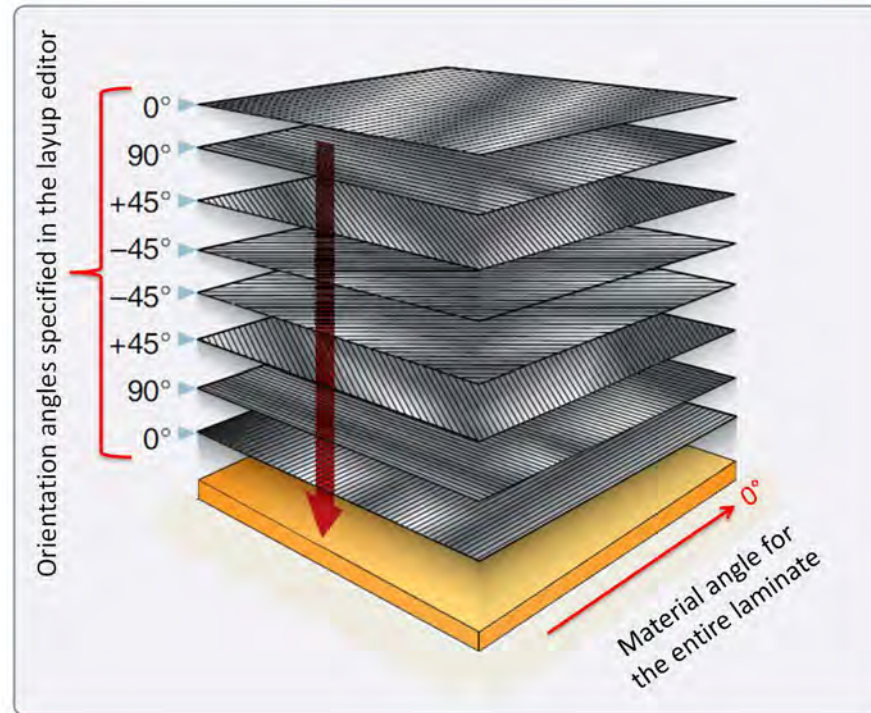
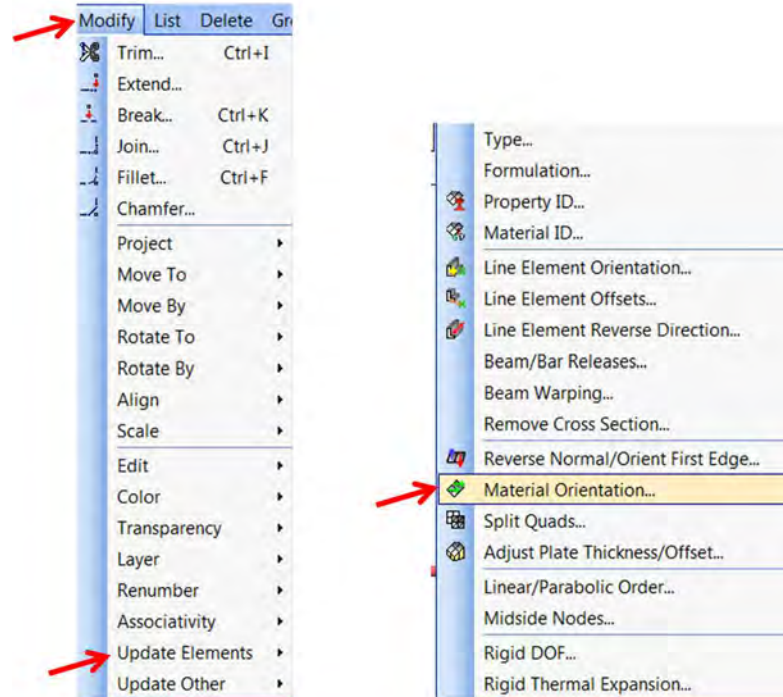


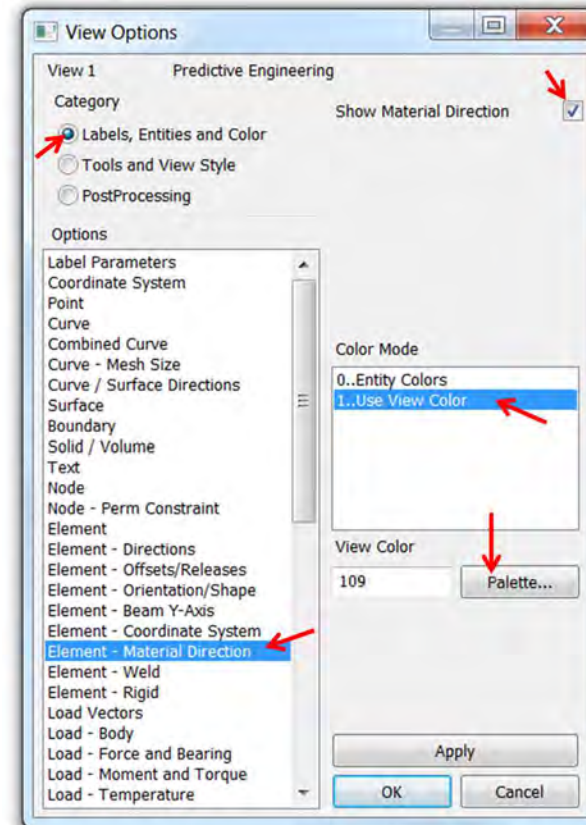
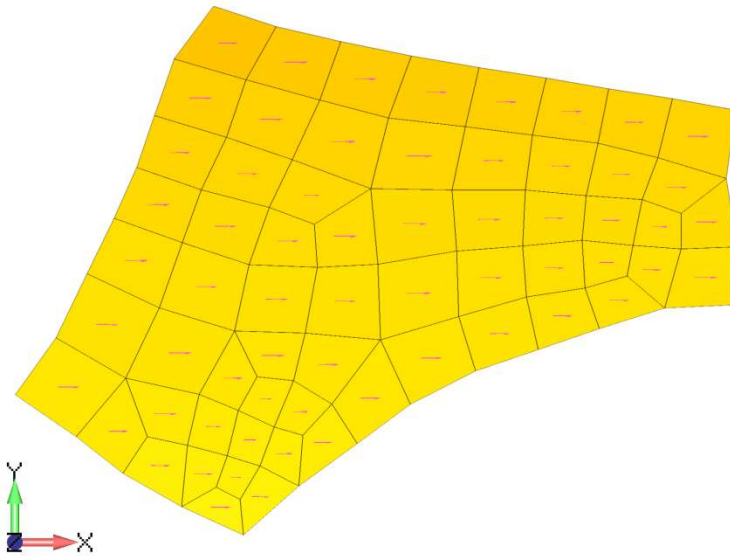
Image source: https://www.faa.gov/regulations_policies/handbooks_manuals/aircraft/amt_airframe_handbook/media/ama_Ch07.pdf

3.5.1 DEFINING A MATERIAL ANGLE ON THE ELEMENT

One can define/update the material angles using toolbar Modify -> Update Elements -> Material Orientation.

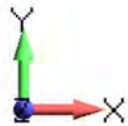
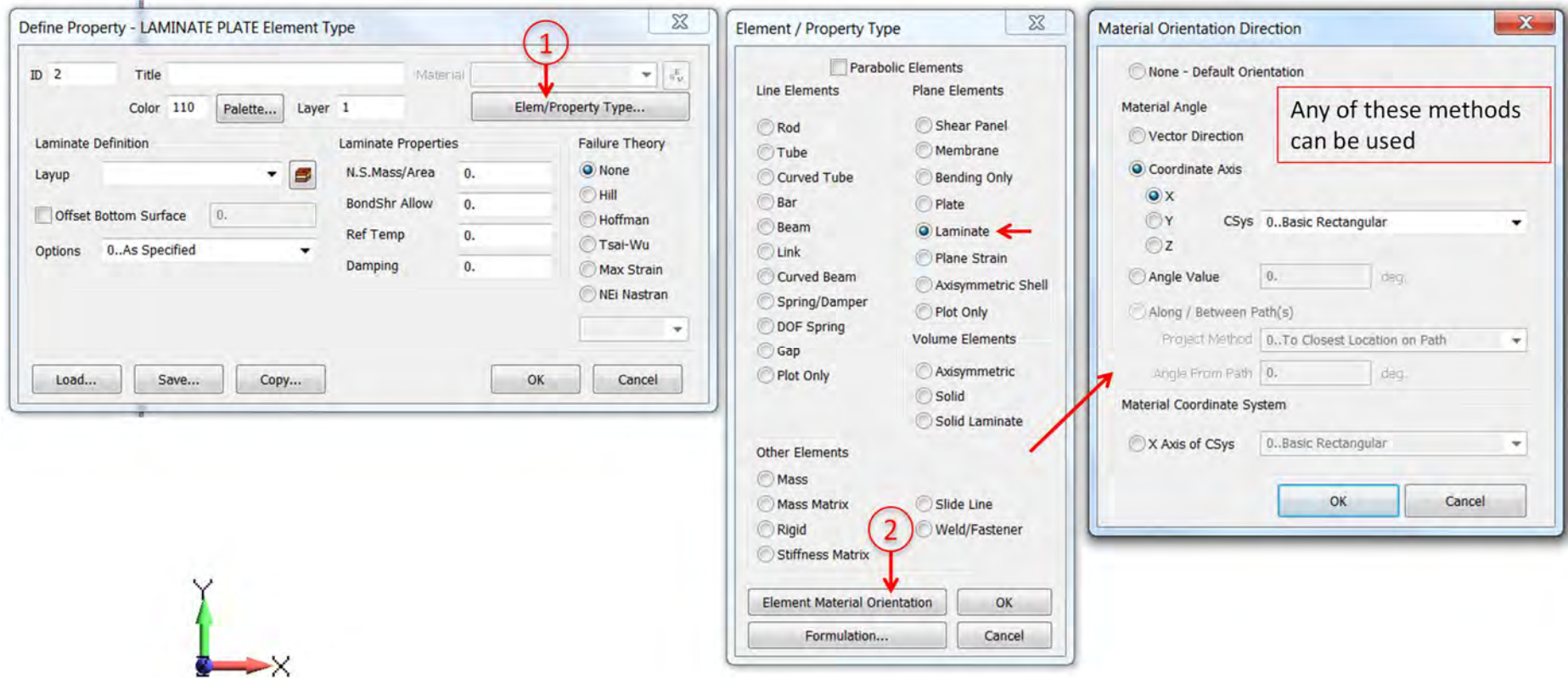


This method allows one to define a material angle to a selected set of elements or all the elements at a time. It is important to note that the material angle is independent of the element coordinate system and the element shape. For example, the image below shows elements that are distorted. However, the material angles are all aligned to the global-x direction. If a material angle is not defined, FEMAP will show an error message. If you miss defining a material angle for a few elements in the model, it can be quite difficult to trace the elements with missing definitions. We have developed an API to filter these elements with missing elements and group them. By accessing this newly created group, one can assign/update the material angles to these elements only. The API is called Composites Material Angle Checker, and can be downloaded from our website at <http://www.appliedcax.com/support-and-training/apis/apis.html>. One can verify the material angles in the laminate elements by selecting the following options under F6 or View Options.



3.5.2 DEFINING A MATERIAL ANGLE IN THE PROPERTY CARD

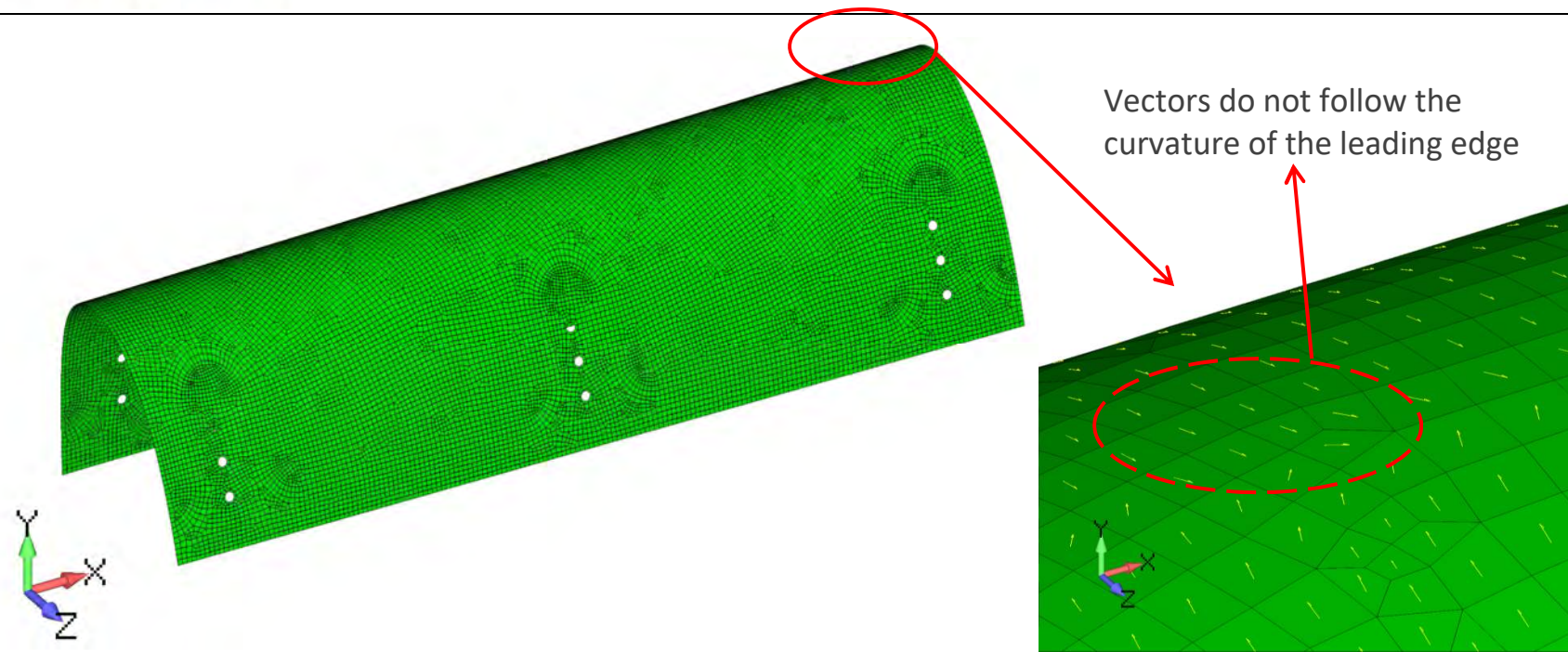
In this approach, we can define a material angle directly in the property card. However, this will assign the material angle to all the elements that the property id is associated with. One can also check the assigned material angles and update them (in case of any incorrect material angles) accordingly using the procedure in approach 1.



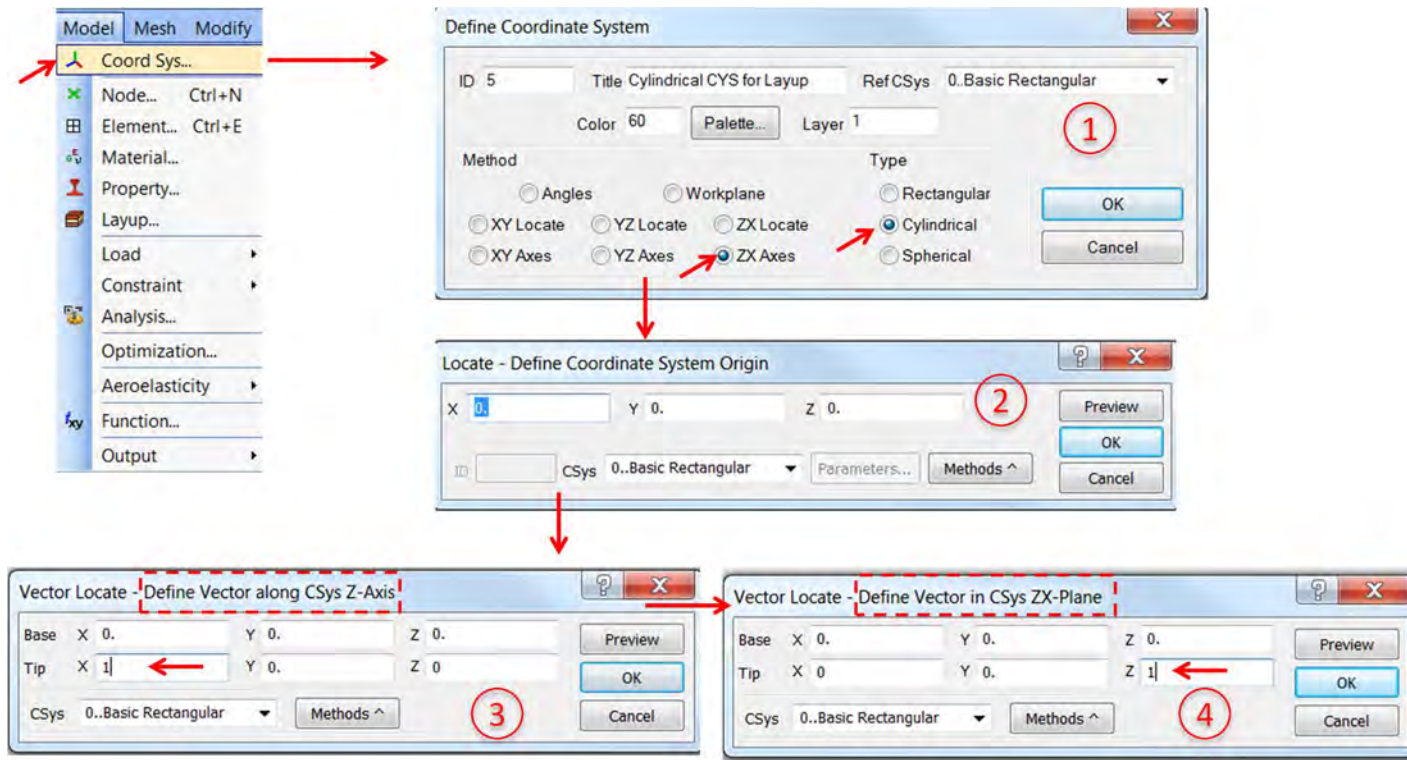
In case of a complex geometry with curved surfaces, material angles can be effectively defined using the cylindrical coordinate system by using the tangent direction if the vector has to follow a curved surface. Always remember to verify your material angles!

3.5.3 ADVANCED APPLICATION EXAMPLE OF ASSIGNING MATERIAL ANGLE TO A COMPLEX GEOMETRY

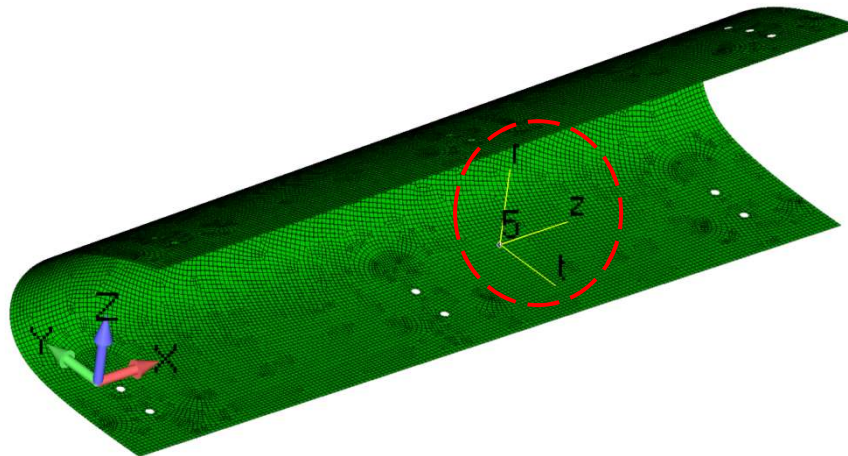
The example below shows a composite wing leading edge. The 0° material angle needs to follow the curvature of the leading edge. However, we can see from the picture that the vectors (representative of the material angles) do not follow the curvature.



In order to assign the material angle accurately, we can use the cylindrical coordinate system (R, T, Z) instead of a rectangular coordinate system (X, Y, Z). We want to create a new cylindrical coordinate system (instead of an existing cylindrical coordinate system) that will do a better job in accurately assigning the material angles. Follow the steps below to create a new cylindrical coordinate system.

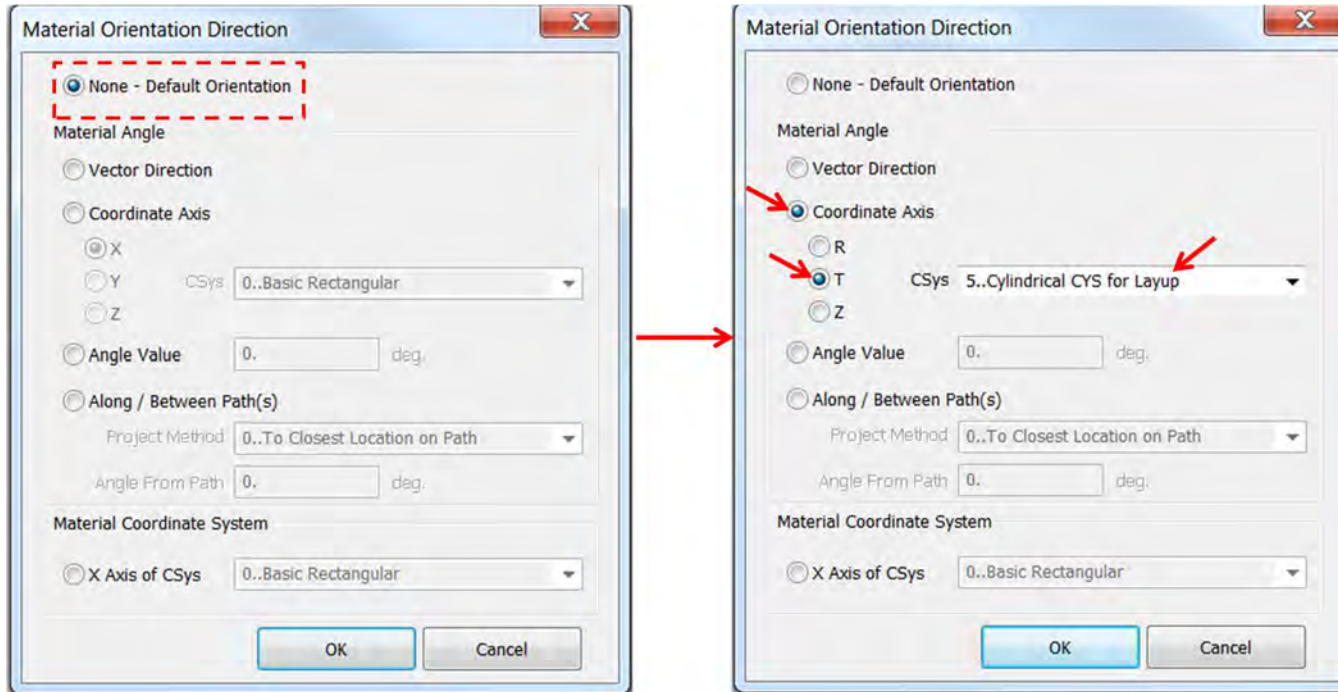


The new coordinate system is shown below.



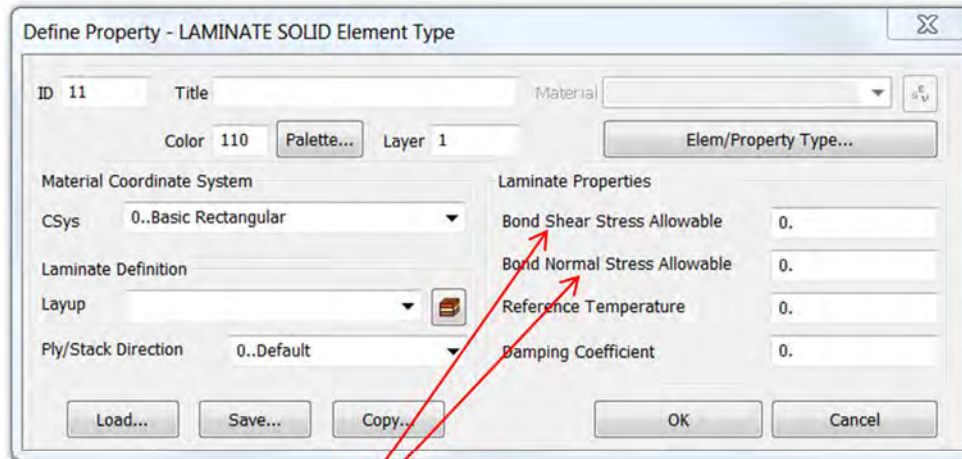
We will use the procedure detailed in section 3.5.1 (Defining a material angle on the element) to update the material angle. Go to toolbar Modify -> Update Elements -> Material Orientation.

This will prompt you to select the elements for which the material angle has to defined/updated. For this example, we will select all the laminate plate elements. Next, you will be prompted to select the material orientation direction (shown below).



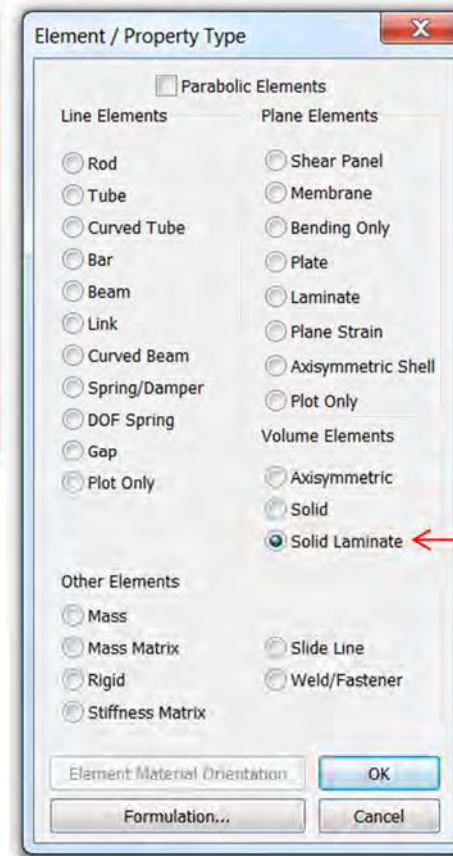
The above procedure will orient the all the material angles along the curvature of the leading edge.

3.6 PROPERTY CARD: 3D LAMINATE MODELING



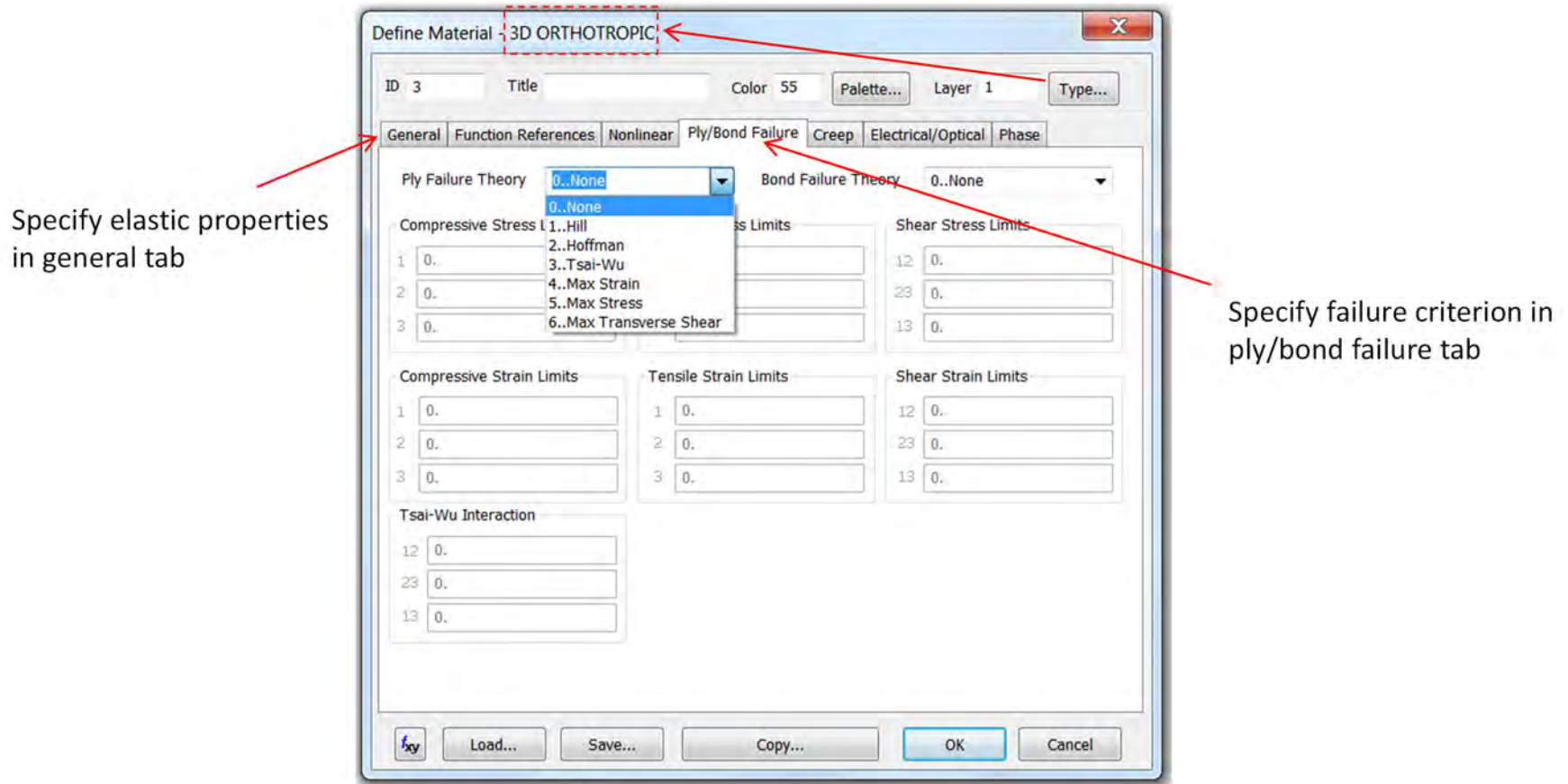
For 3D laminate modeling, one can specify both shear stress allowable and normal stress allowable for the bond.

Notice that, failure theories are not available in the property card. In solid laminates, failure criterion is specified using the materials card.



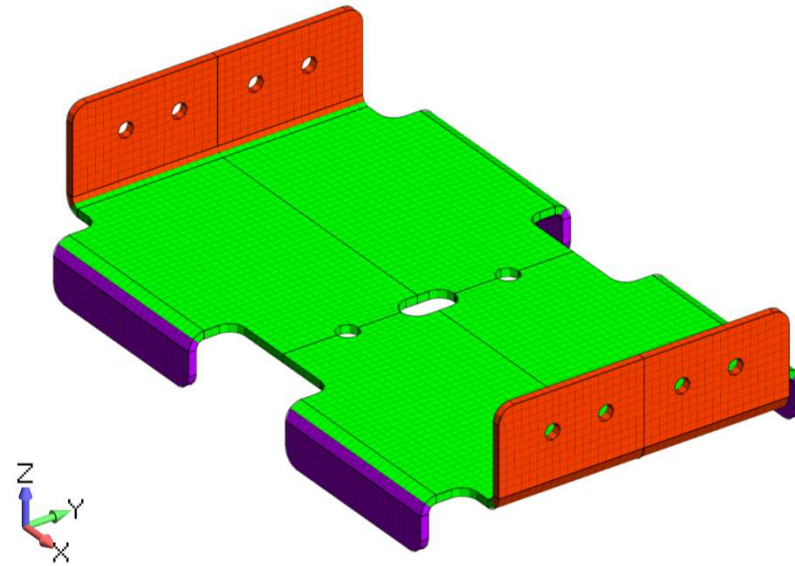
3D laminate modeling

In 3D laminate modeling, we define a ply/stack direction instead of a material angle. For example, if the material angle is in global-x and the layup stacking in global-z, then we specify the ply/stack direction as XZ (13). Unlike 2D, we cannot use approach 1 (Modify -> Update Elements -> Material Orientation) for 3D solid laminate elements. We can create multiple property cards (if necessary) with different ply/stack directions for sections of your geometry. And try to explore other coordinate systems for specifying the ply/stack direction.

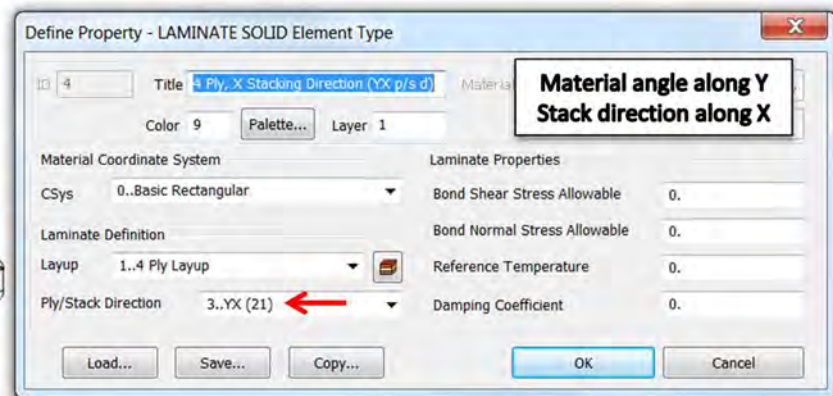
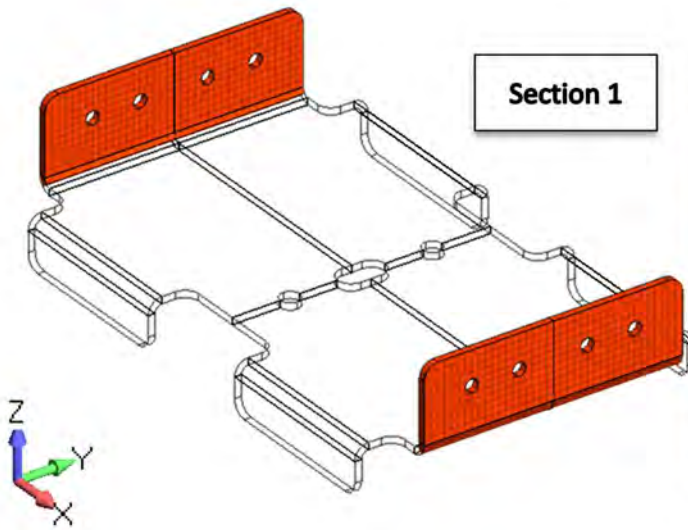


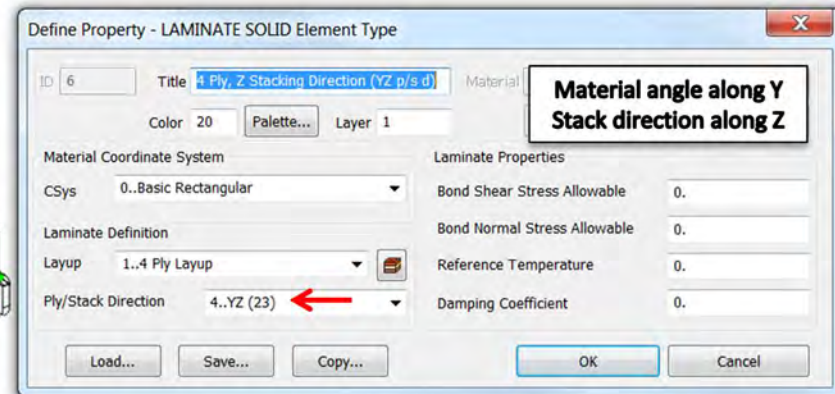
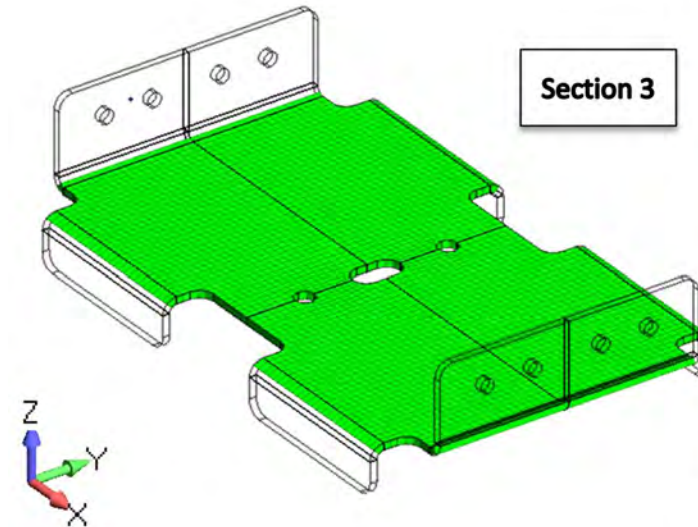
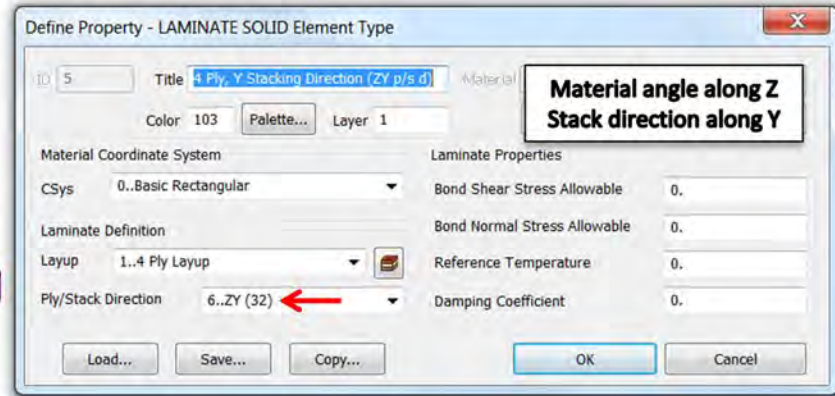
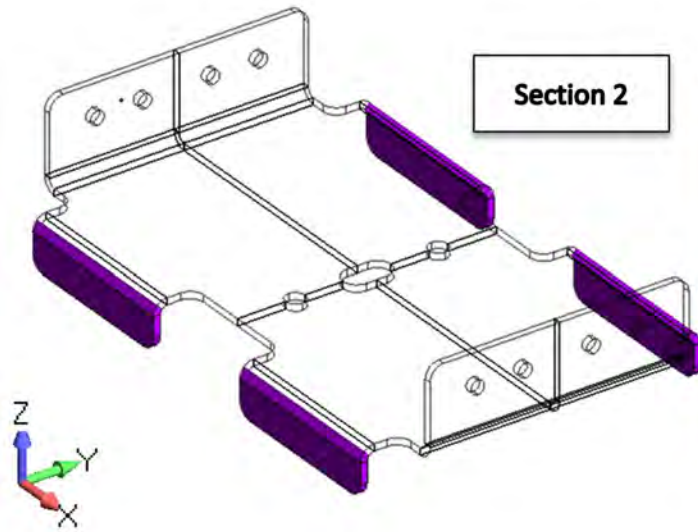
3.7 MULTI-MATERIAL ANGLE EXAMPLE OF A 3D COMPOSITE MODEL

This example shows a composite component with sections oriented in different directions. The component is modeled using laminate solid elements. Here, we cannot use a single ply/stack direction for all the sections of the component.



In this example, we will create three property cards (identified by color as shown above) to define ply/stack direction separately for each of the three sections of the composite component. The following images will show how ply/stack directions have been defined for these sections.





4. EXAMPLE 1: CREATING A 2D LAMINATE MODEL IN FEMAP

4.1 INTRODUCTION

In this example, we will analyze a simple composite laminate with a hole subjected to uniaxial tension. Composite laminates are often studied for open-hole tension in the aerospace industry. The material, layup, and the thicknesses used in this example are from a real world composite part. Here, we will study two types of layups with the same number of layers and same orientations but stacked in different ways. We will analyze the differences in the results just by altering the layup stacking order. We will learn this unique behavior of composites by working out this example. With multiple plies in the layup, this is also a good example for post processing.

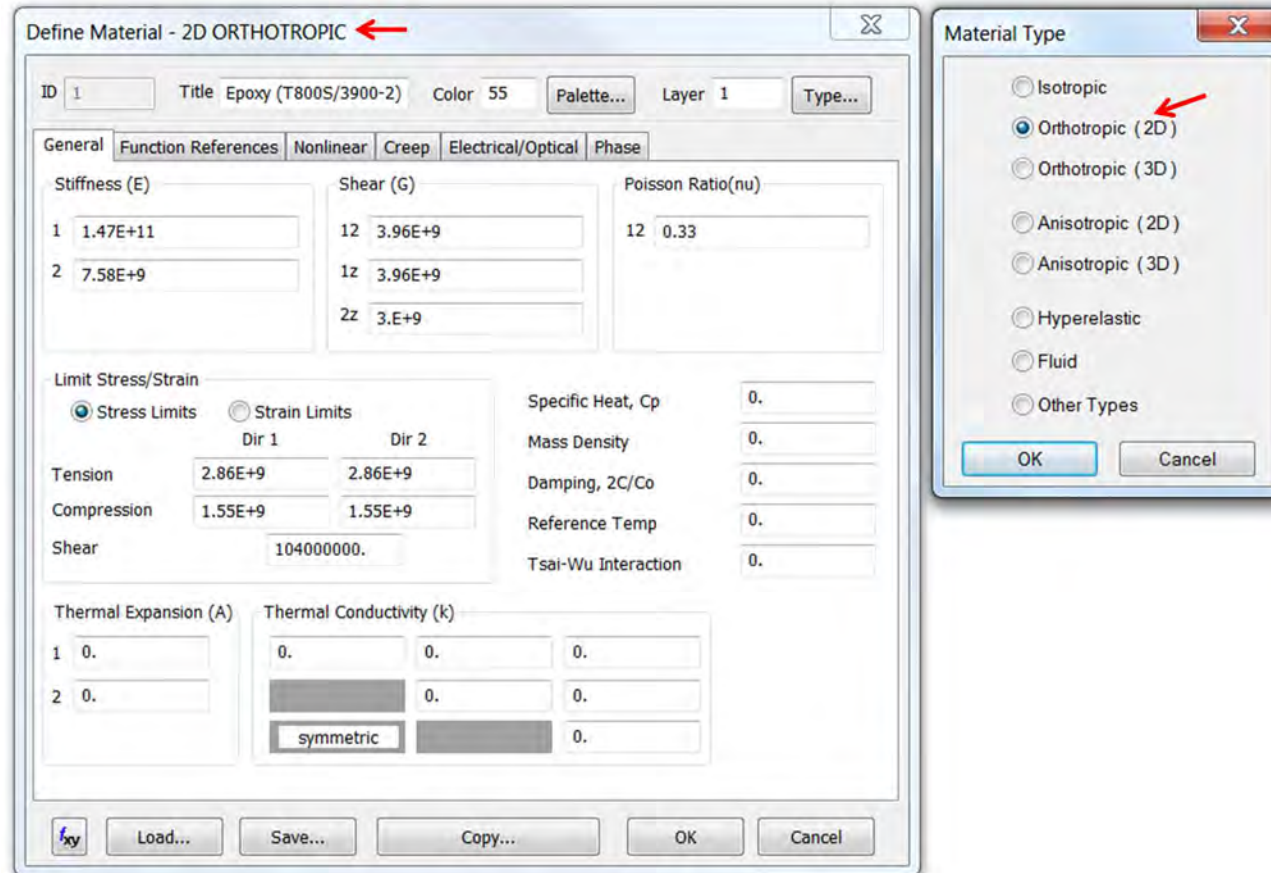
4.2 CREATING THE MATERIAL PROPERTY

The material properties used for the laminate correspond to a unidirectional carbon/epoxy lamina. Accordingly, the material type needs to be '2D Orthotropic' for this problem. The material properties are shown in Table 1.

Table 1: Mechanical properties of a unidirectional T800S/3900-2 lamina

Property	Unit	T800S/3900-2
Longitudinal elastic modulus (E_1)	GPa	147
Transverse elastic moduli ($E_2 = E_3$)	GPa	7.58
In-plane shear modulus (G_{12}) = Interlaminar shear modulus (G_{13})	GPa	3.96
Interlaminar shear modulus (G_{23})	GPa	3.00
In-plane Poisson's ratio (ν_{12}) = Interlaminar Poisson's ratio (ν_{13})		0.33
Interlaminar Poisson's ratio (ν_{23})		0.38
Longitudinal tensile strength (X_t) = Longitudinal compressive strength (X_c)	GPa	2.86
Transverse tensile strength (Y_t) = Transverse compressive strength (Y_c)	GPa	1.55
Shear strength (S)	GPa	0.104

Since the laminate model is 2D, the direction 3 properties can be ignored for this problem.



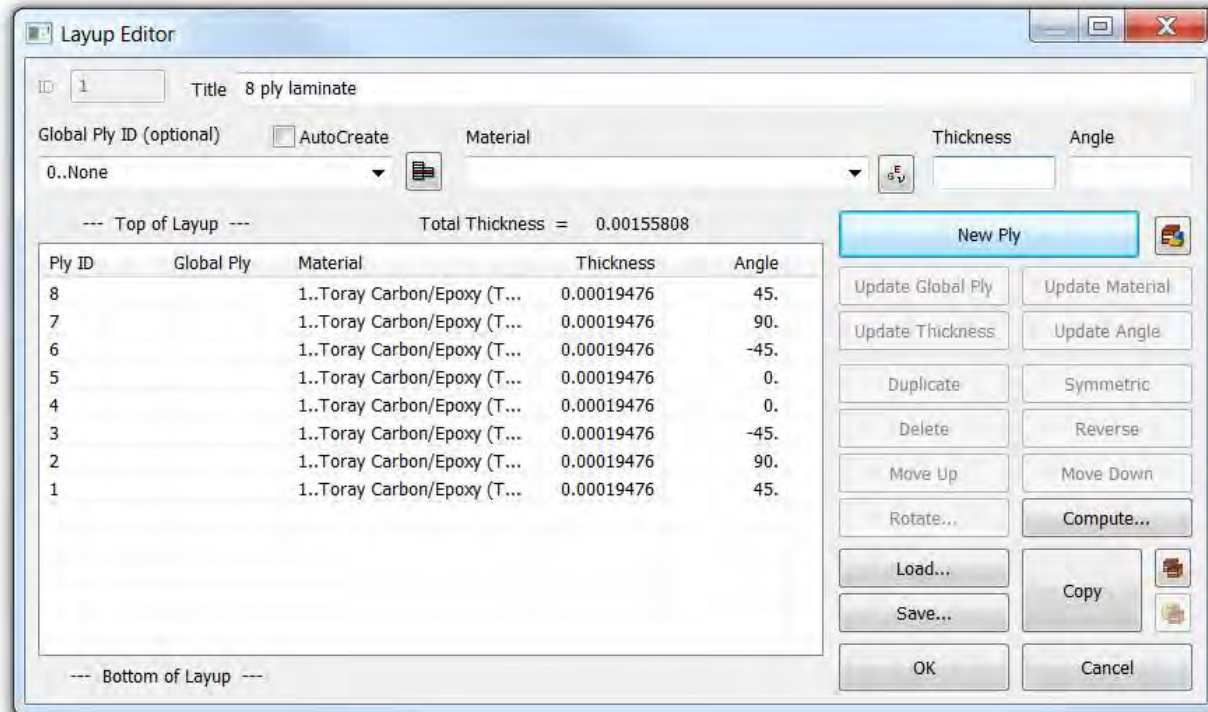
4.3 DEFINING THE LAYUP

We will analyze two types of layups in this model. The idea is to observe how the layup can affect your analysis even if the laminate effectively has the same number of layers, same orientation angles, but stacked up in a different sequence.

Layup 1: $[45^\circ/90^\circ/-45^\circ/0^\circ]_s$ with each layer at 0.195 mm thickness. The subscript 's' in the layup definition shows that it is symmetric laminate. This implies that the layup configuration is $[45^\circ/90^\circ/-45^\circ/0^\circ/0^\circ/-45^\circ/90^\circ/45^\circ]$.

Layup 2: $[45^\circ/90^\circ/-45^\circ/0^\circ/45^\circ/90^\circ/-45^\circ/0^\circ]$. Layup 2 has 8 layers as layup 1 except that the layup configuration is no longer symmetric about laminate’s mid-plane.

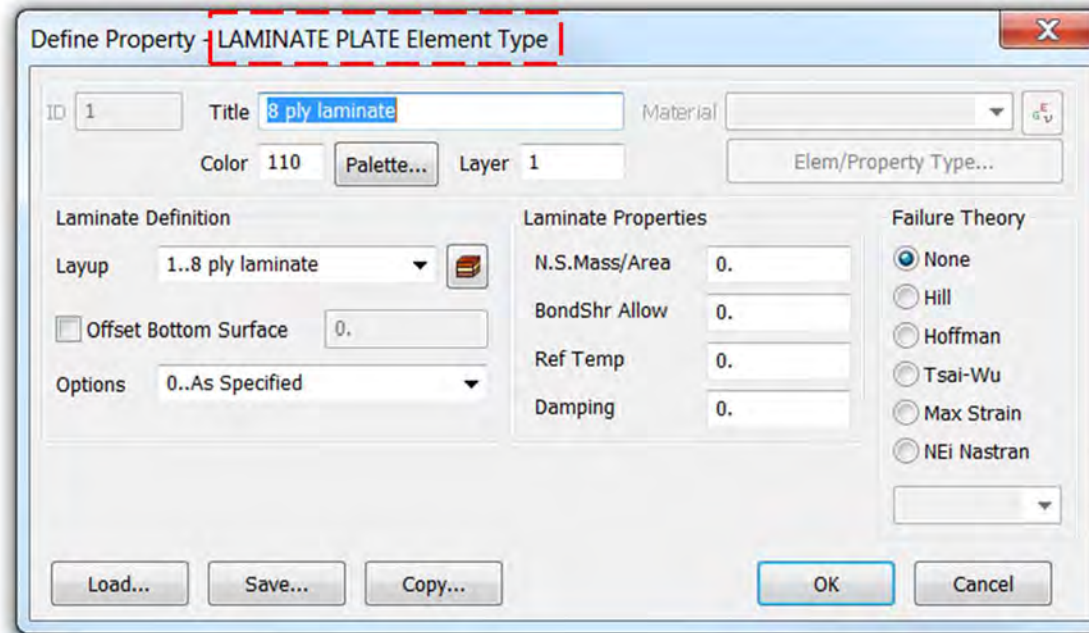
The layup editor with Layup 1 defined is shown below. Layup 2 can be defined in a similar manner by changing the orientation angles.



4.4 DEFINING THE LAMINATE PROPERTY (PCOMP)

The Nastran property card for 2D laminates is PCOMP. Detailed information on the PCOMP card is available in the Nastran User Guide. The 2D laminate formulation is based on classical lamination theory. Select the ‘Laminate’ element/property type in the property card. By default, FEMAP assigns ‘plate’ element type for 2D models. The layup defined in the previous step can be assigned in the property card. Since we have defined all the 8 layers in the layup, ‘options’ is left to its default ‘As Specified’ in the property card. Alternatively, we can define 4 layers only in the layup and use ‘Symmetric’ option. However, this option has to be used with caution as it cannot be used for unsymmetric

laminates. Ideally, one can define all the layers in the layup editor unless there are a large number of layers (e.g., thick composite).

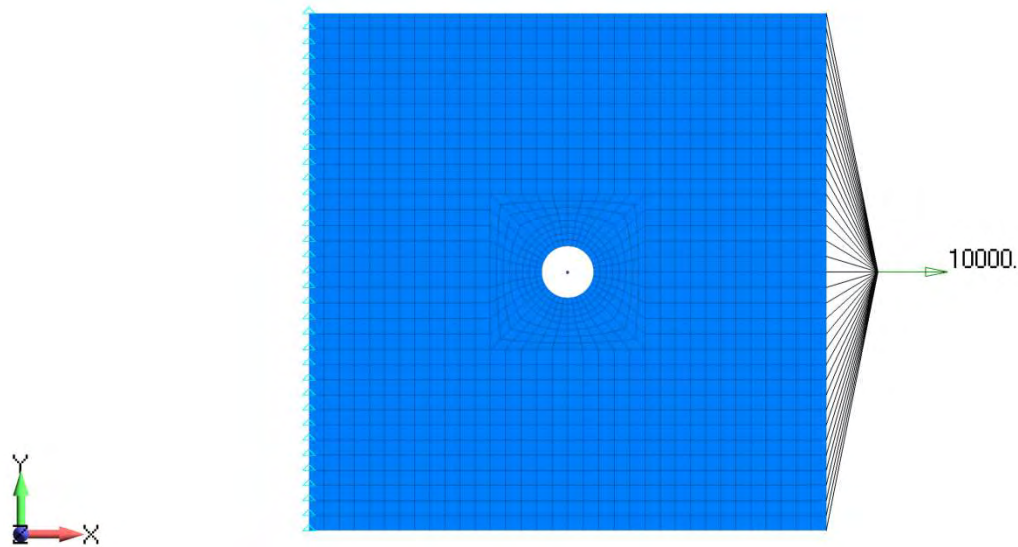


4.5 SPECIFYING MATERIAL ANGLES

For this problem, we will use global x-direction as our material angle. We can use one of the procedures which were introduced earlier.

4.6 ANALYSIS SETUP AND POST PROCESSING

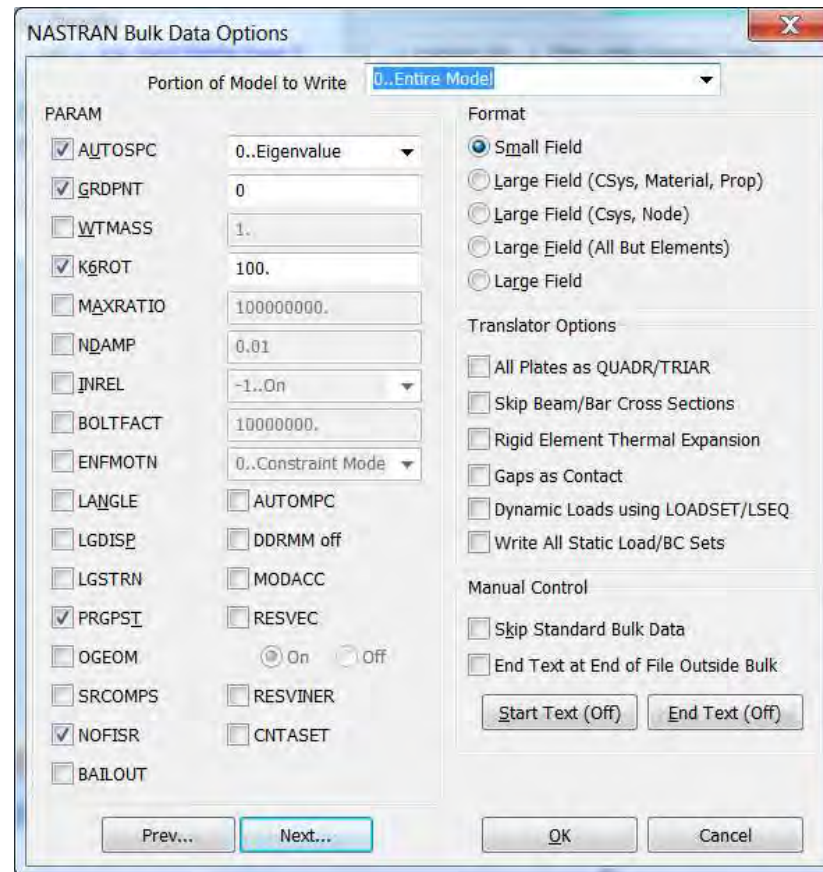
For this model, a force of 10 kN is applied on the nodes on one end and fixed constraints are applied on the other end as shown below. To simulate the experimental loading behavior, the 10 kN load is applied on an independent node and this node is rigidly linked to nodes on the right end of the laminate model (shown below). When the load is applied on the independent node, the resulting displacement is translated to the dependent nodes.



The model is ready to be analyzed and is solved for a linear static case. The model is analyzed for the two layups and the deformations observed in both these cases are discussed.

4.6.1 ANALYSIS SET MANAGER SETUP FOR COMPOSITES

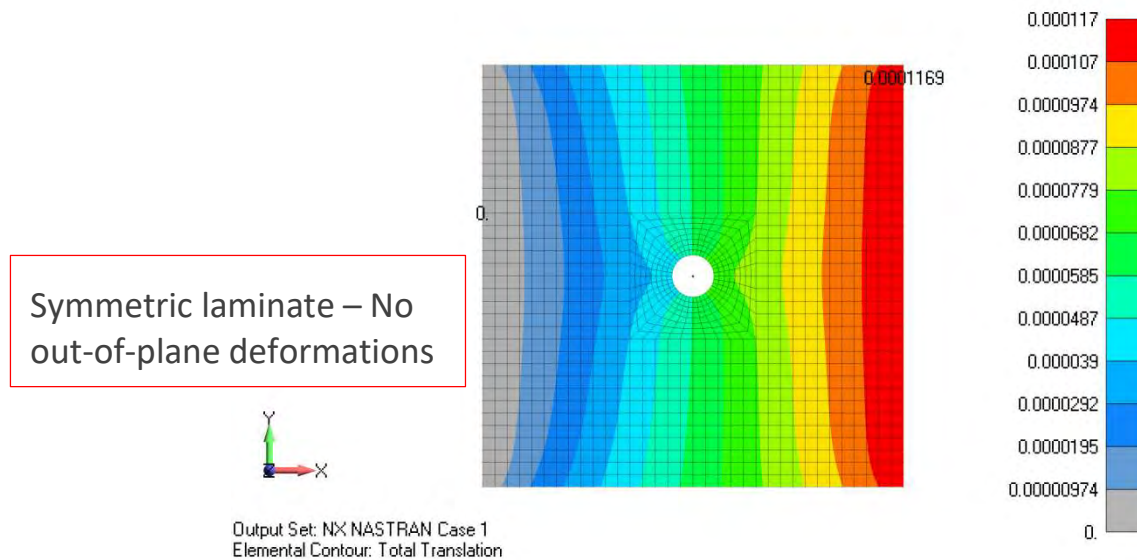
- SRCOMPS controls the computation and printout of ply strength ratios. When 'on', ply strength ratios are output for composite elements that have failure indices requested.
- NOFISR controls the printout of the composite failure indices and strength ratios. When 'on', the failure indices and strength ratios will not be printed.

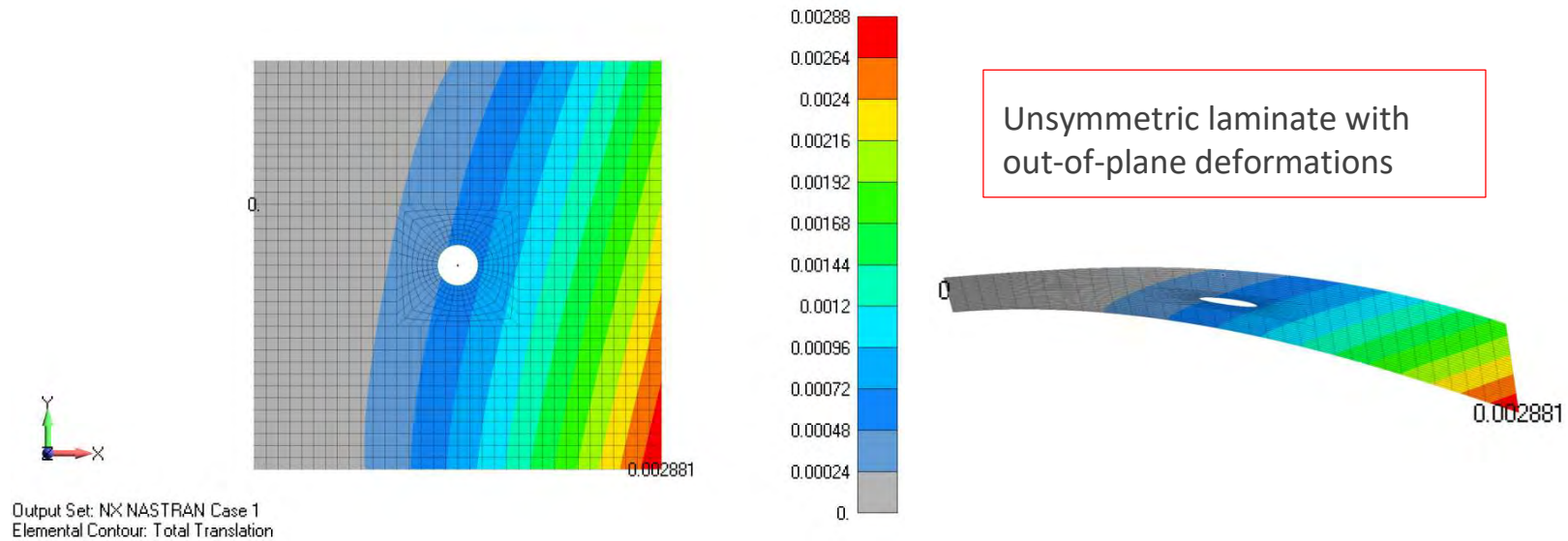


4.6.2 POST PROCESSING THE RESULTS

Initially, the model with the symmetric laminate ($[45^\circ/90^\circ/-45^\circ/0^\circ/0^\circ/-45^\circ/90^\circ/45^\circ]$) was analyzed. The total translation in this model is shown below. The deformations are symmetric about the x-axis. Under the applied uniaxial tensile load, deformations were inplane as expected. No out-of-plane deformations were observed.

Next, the model with unsymmetric laminate ($[45^\circ/90^\circ/-45^\circ/0^\circ/45^\circ/90^\circ/-45^\circ/0^\circ]$) was analyzed. From the contours (shown below), it can be observed that the deformations were not symmetric about the x-axis. Also, we can now see that the unsymmetric laminate resulted in an out-of-plane deformation although the loading was uniaxial.





This kind of behavior is unique in fiber-reinforced composite laminates and often not explored due to inherent design challenges. An unsymmetric layup can result in warping of the laminate as early as during the manufacturing process when the laminate is cured at high temperatures. Thus, it is quite challenging to design an unsymmetric laminate to suit a specific loading environment. One practical example of an unsymmetric laminate application is in the forward swept composite wings of the X-29 aircraft [2].

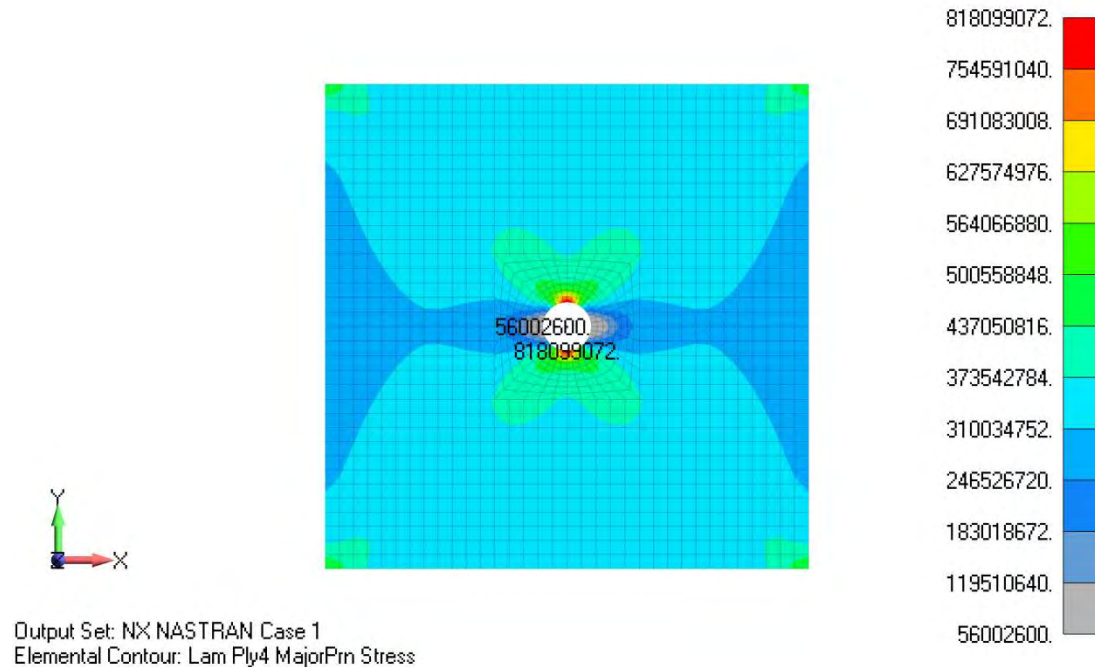
X-29 aircraft



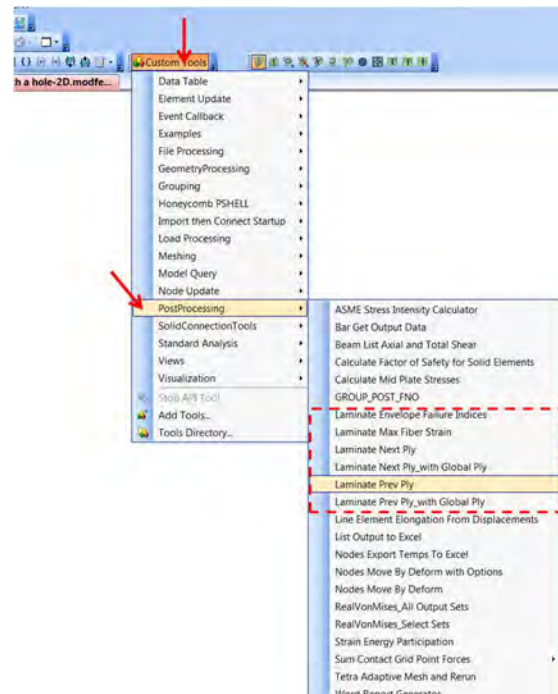
Photo courtesy of Structural Mechanics Corporation

<http://structuralmechanics.com/about/resources/newsletter/articles/you-pull-it-twists-tailored-composites/>

If this subject interests you, please see the Appendix. Next, we proceed to stress distributions in the individual plies. For this discussion, we will use the symmetric laminate results.



The stress contour shown above corresponds to the major principal stress in ply 4 (0° layer). Similarly, we can plot stress distribution in other layers. FEMAP has bunch of output options and sometimes it can be difficult to find the same output vector for another ply. There are some custom tools available in FEMAP which can be effectively used for laminates. For example, we can use the ‘Laminate Prev Ply’ tool to plot ply 3 Major Principal Stress (provided we have a plot of ply 4 Major Principal Stress) without putting effort in finding the output vector amongst large set of results. This will be particularly useful if you have a large number of layers in the laminate. Some of the tools which are specific to laminates are shown below. These tools can be accessed from Custom Tools -> PostProcessing.



If you would like to use these API's frequently, we can create a customized toolbar with these commands. It is easier this way to find a command in the toolbar rather than searching for these commands in the Custom Tools. We have created a video on creating a customized toolbar and can be accessed from https://www.youtube.com/watch?v=RT_7ulor6Ww

5. EXAMPLE 2: CREATING A 3D LAMINATE MODEL IN FEMAP

5.1 INTRODUCTION

In this example, we will develop a 3D laminate model to analyze a simple composite laminate with a hole subjected to uniaxial tension. This is an extension of example 1 to a 3D laminate model. The goal is to understand the applicability of both the models. The material, layup, thicknesses, and the approach used in this example are same as in the example 1. A major difference between the 2D laminate model and the 3D laminate model is that the former will not take out-of-plane stresses into account while the latter will. The 3D laminate models will be useful if you are modeling a composite structure with free edges like holes where the out-of-plane shear stresses can be important.

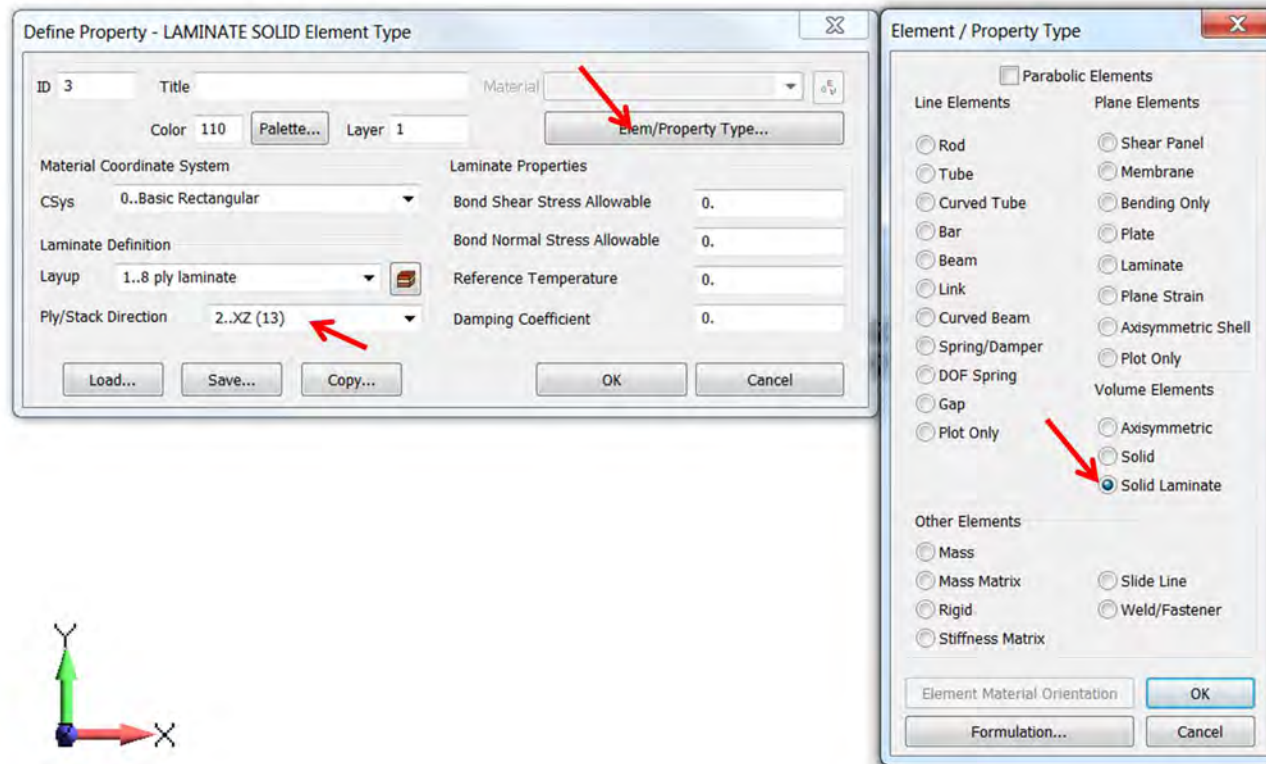
5.2 CREATING THE MATERIAL PROPERTY

Defining the material properties for a 3D laminate model is similar to the 2D model except that the material type is now 3D orthotropic. Accordingly, direction-3 properties should also be defined. All the required properties are defined in Table 1.

5.3 DEFINING THE LAYUP

Specifying the layup configuration is similar to the 2D case. However, the material property (3D orthotropic) created in the previous step has to be used instead of 2D orthotropic properties as in example 1.

5.4 DEFINING THE LAMINATE PROPERTY (PCOMPS)



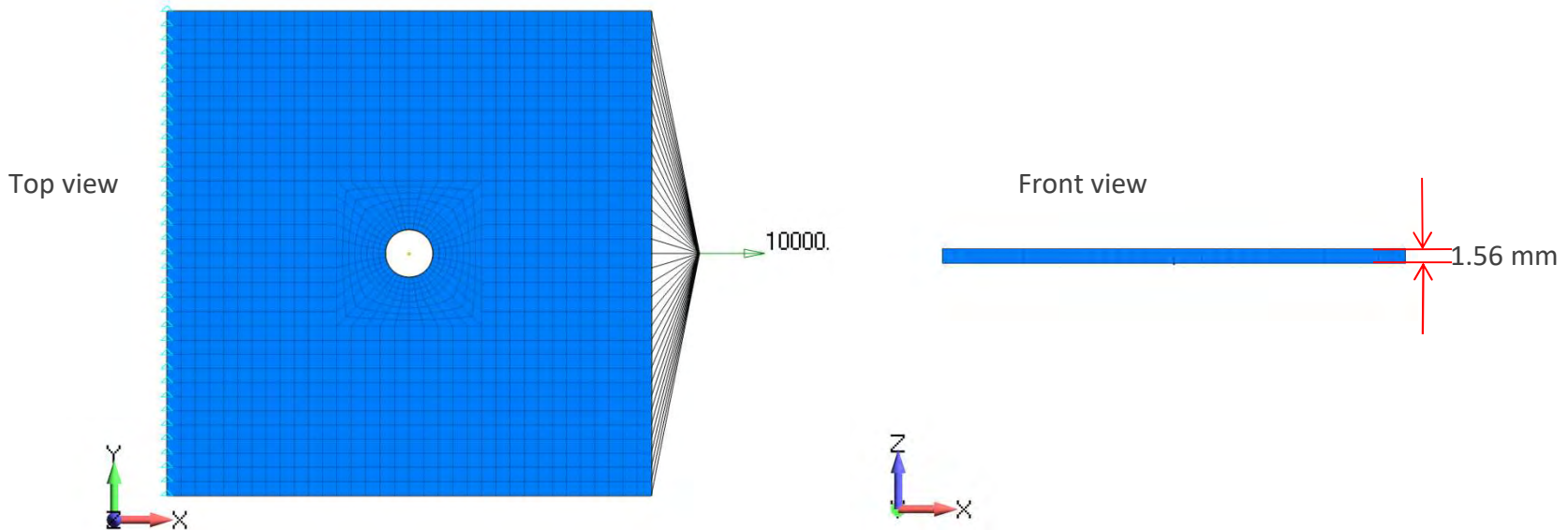
The Nastran property card for 3D laminates is PCOMPS. Detailed information on the PCOMPS card is available in the Nastran User Guide. Select the ‘Solid Laminate’ element/property type in the property card. By default, FEMAP assigns ‘Solid’ element type for 3D models. The layup defined in the previous step can be assigned in the property card. The interface for the 3D laminate property card is slightly different from the 2D case. Here ‘Ply/Stack Direction’ has to be assigned. For example, for the 3D laminate model the Ply/Stack Direction is specified as ‘XZ (13)’. Here, ‘X’ corresponds to the direction of the material angle (as defined in 2D case) and ‘Z’ corresponds to the stacking direction. Unlike the 2D laminates, the 3D laminate formulation is not based on classical lamination theory.

5.5 SPECIFYING MATERIAL ANGLES

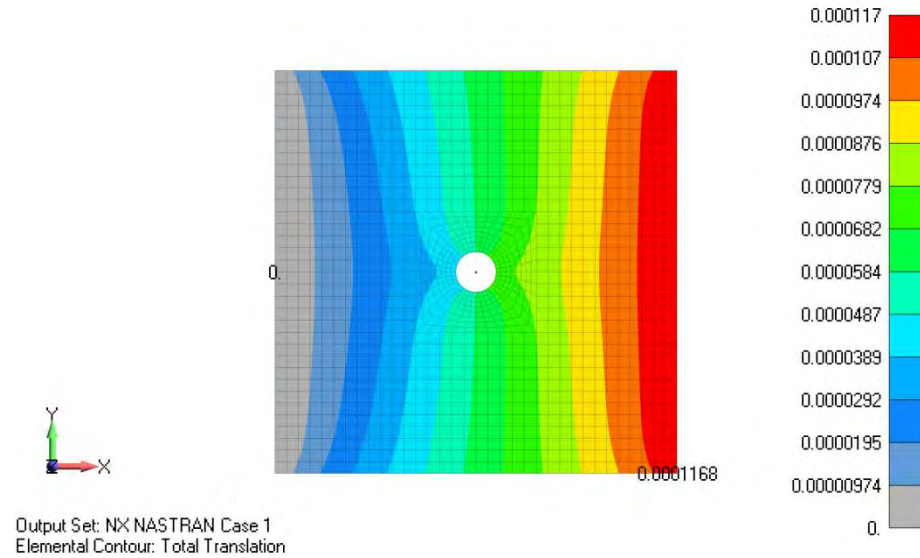
The only difference between specifying material angles for a 2D case and 3D case is that the 3D model requires specifying the ‘Ply/Stack Direction’. The overall concept is still the same.

5.6 POST PROCESSING THE RESULTS

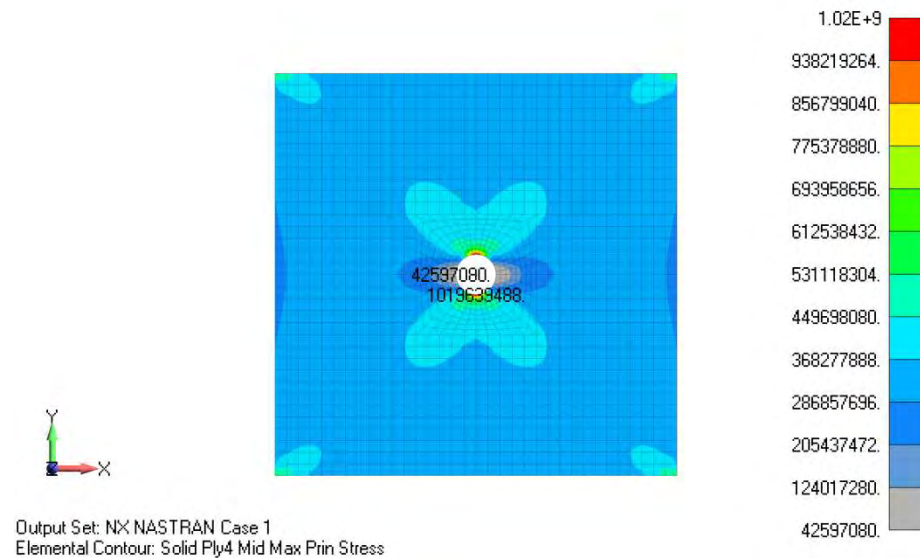
The image below shows the finite element mesh of the 3D laminate model. In this example, only one solid element is defined through the thickness. The thickness of the element corresponds to the total thickness of the laminate – 1.56 mm.



The 3D laminate model is analyzed with the symmetric layup configuration. The observed displacements and stresses are shown below. The displacement results are quite similar to the 2D case.



The major principal stress distribution in ply 4 (0° layer) is shown below.



6. EXAMPLE 3: MODELING A SANDWICH COMPOSITE

6.1 INTRODUCTION

In this example, a 3D sandwich composite model is analyzed for a uniform pressure loading. The sandwich composites have a core (e.g., honeycomb, foam, etc.) sandwiched between two facesheets. Typically, the facesheets carry the majority of the inplane and bending loads while the core takes shear. A sandwich composite can be conveniently modeled using the 2D laminate layup by defining the core as one of the layers in the layup editor (shown at the end of this example). While this procedure is easy, interlaminar shear stresses (around free edges) become important in sandwich composites and 2D laminate models (based on classical lamination theory) do not account for the out-of-plane stresses.

6.2 CREATING THE MATERIAL PROPERTY

Two material models, each for the composite facesheet and core are defined. The composite facesheet is modeled as a 3D orthotropic material and the properties are shown in Table 1 (example 1). The core is modeled as an isotropic material with $E = 4 \text{ GPa}$ and $\nu = 0.25$ [3].

6.3 DEFINING THE LAYUP

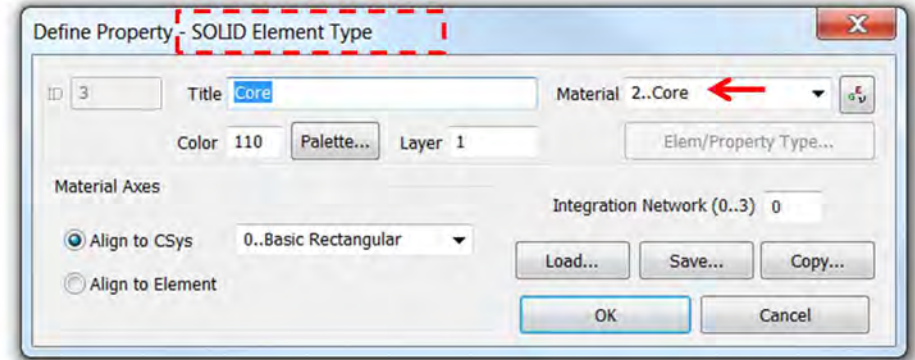
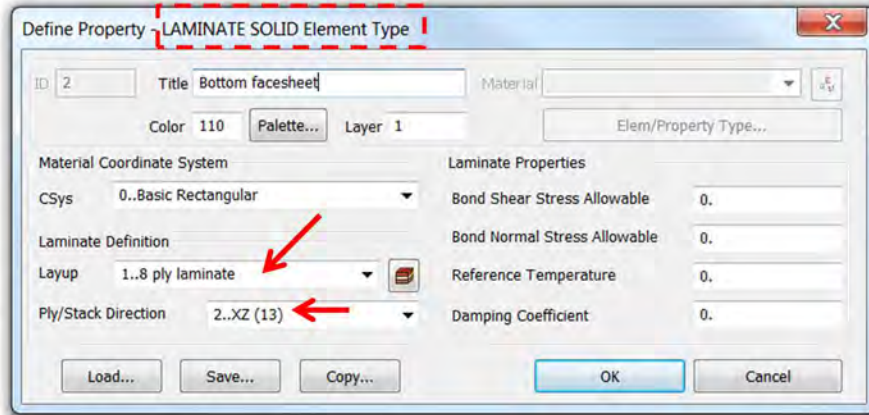
Defining the laminate layup for the facesheets is same as in example 2. However, it should be noted that we have two facesheets and so we have to create two solid laminate property cards, one for the top facesheet and one for the bottom facesheet. If two layups are not created separately for each of the facesheets, we will see output vectors corresponding to 8 plies only in the results set. If two layups are created, then we can see output vectors for 16 plies, 8 for the bottom facesheet and 8 for the top facesheet.

6.4 DEFINING THE PROPERTY CARDS

Three property cards have to be created, two of solid laminate type for facesheets and one solid type for the core. Each of the solid laminate property cards have to be assigned to the corresponding facesheets. The material angles specification for the solid laminates is similar to the previous example.

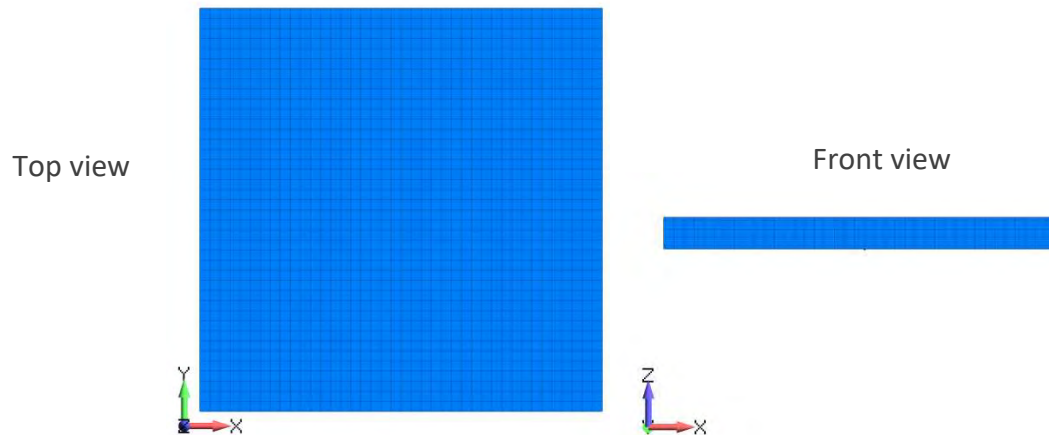
Facesheet

Core

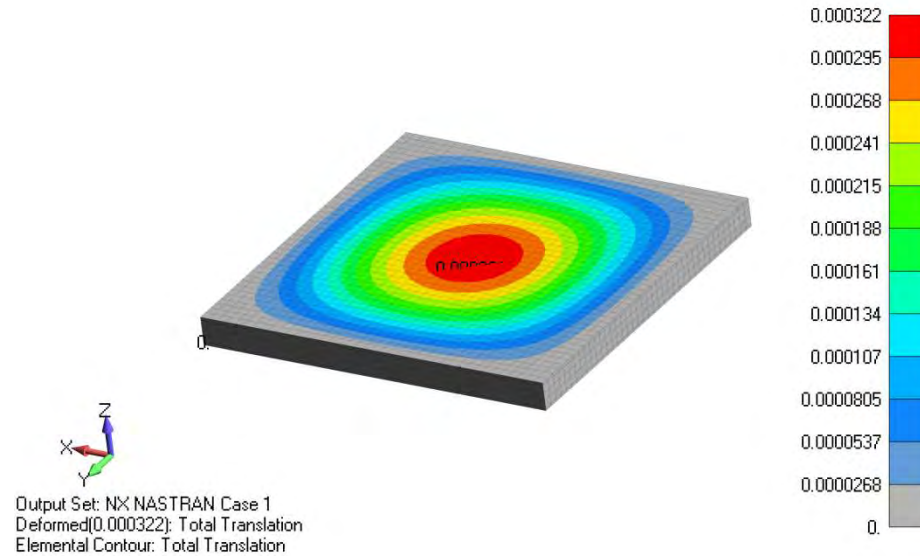


6.5 POST PROCESSING THE RESULTS

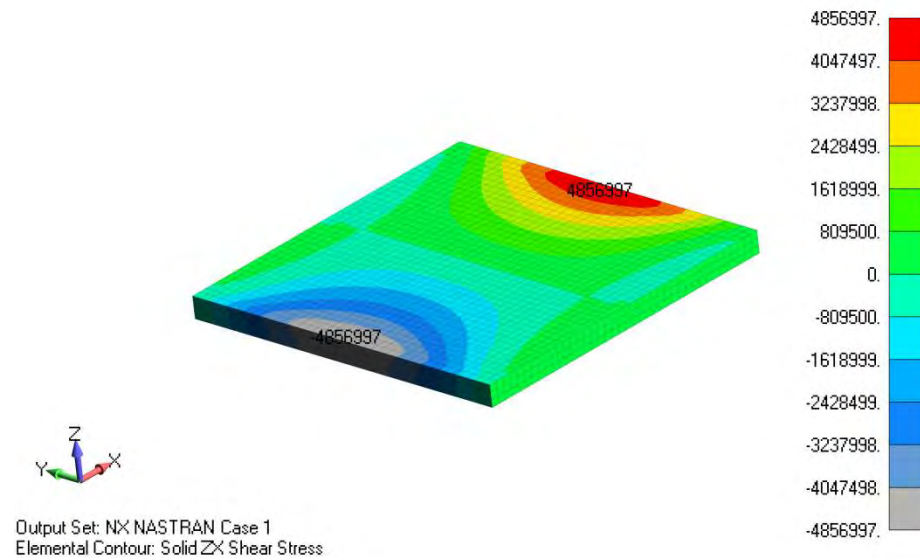
Clamped boundary conditions have been applied on the sandwich composite edges and a pressure load of 1 MPa is applied on the top surface.



The deformation plot of the sandwich composite due to the applied pressure load is shown below.



The interlaminar shear stress distribution in the core is shown below. One can mask the facesheet elements while plotting the shear stress distribution in the core.

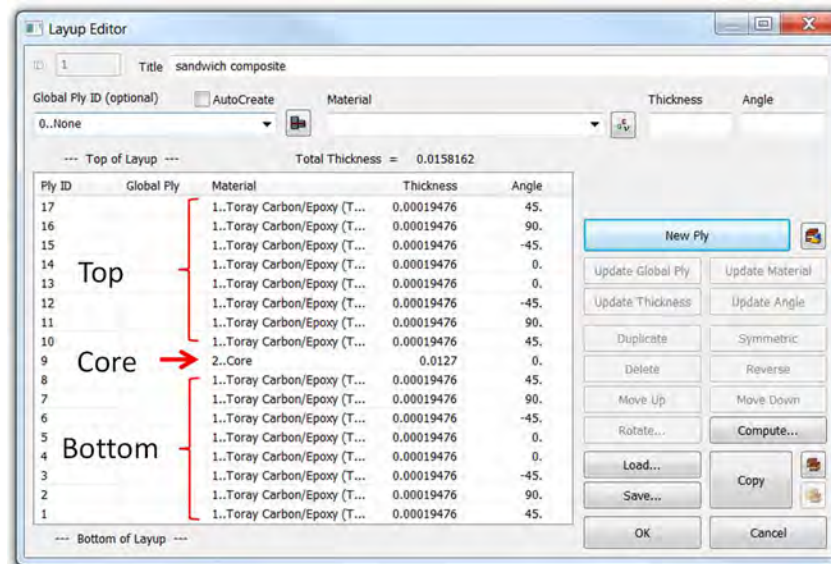


The bond between the core and facesheets is one of the critical regions for delamination in sandwich composites. These interlaminar shear stresses are higher at the free edges and these areas are potential regions for delamination initiation. One can compare the interlaminar shear stresses to the core-to-facesheet bond shear allowable and analyze for any possible delaminations.

6.6 OTHER METHODS FOR SANDWICH COMPOSITE MODELING

In this example we have developed a 3D sandwich composite model in which the facesheets and the core are modeled as solids. Other methods by which the above problem can be analyzed are:

1. 2D sandwich composite modeling in which the facesheets and the core are all 2D and can be defined in a single layup definition. This is a convenient approach however, as discussed earlier, the 2D laminate models are based on classical lamination theory and do not account for the out-of-plane stresses. Secondary methods are used to estimate the interlaminar stresses.



2. A mix of 2D laminates and 3D core. In this method, the facesheets can be modeled as 2D and the core can be modeled as a solid. One should be careful about defining the laminates for this configuration. Both the facesheets have to be placed at an offset of half the laminate thickness from the solid core.

3. Using the classical plate theory to model sandwich composites. This model requires some hand calculation to be done and is complex as compared to the 2D laminate model. The Nastran property card corresponding to the plate model is the PSHELL card. In the method 1 described above, Nastran converts a PCOMP property into an equivalent PSHELL. So, both method 1 and method 2 should give similar results if all the properties are accurately defined. To avoid confusion, method 1 is preferred over method 2 as both are the same in terms of how Nastran interprets the property card. Alternatively, FEMAP has a custom tool to define a sandwich composite using the PSHELL method. One can do hand calculations and compare their values with the FEMAP calculated value as shown below. This custom tool for sandwich composites can be accessed from Custom tools -> Honeycomb PSHELL -> Honeycomb PSHELL Property.

Honeycomb Panel Cross Section (PSHELL) Property Input

Property Title: Sandwich Composite

Total Face Sheet Thickness ($T = \sqrt{2} \cdot 2$ in figure): 3.12e-3

Face Sheet Material: 1..Toray Carbon/Epoxy (T80)

Core Thickness (D in figure): 12.7e-3

Core Material: 2..Core

Core Density (Rho): 50

Diagram labels: Facesheets, Honeycomb Core, $t/2$, d

Define Property - PLATE Element Type

Title: Sandwich Composite

Material: 1..Toray Carbon/Epoxy

Color: 110

Layer: 1

Property Values:

- Thicknesses, Tavg or T1: 0.00312
- blank or T2: 0.
- blank or T3: 0.
- blank or T4: 0.
- Nonstructural mass/area: 0.635

Additional Options:

- Bend Stiffness, $12I/T^{**3}$: 49.70722
- TShear/Mem Thickness, ts/t : 4.070513
- Bending: 1..Toray Carbon/Epoxy (T)
- Transverse Shear: 2..Core
- Memb-Bend Coupling: 0..None - Ignore
- Mean Dilatational Formulation

Stress Recovery (Default=T/2)

- Top Fiber: 0.00713
- Bottom Fiber: -0.00713

Buttons: Load..., Save..., OK, Copy..., Cancel

Compare your hand calculations to these Femap calculated values

7. LAMINATE FAILURE THEORIES IN FEMAP

7.1 INTRODUCTION

The behavior of composites is complex (as a result of heterogeneous properties) when compared to monolithic materials. Understanding the behavior of composites under extreme conditions of mechanical loading, temperature, and other environmental factors poses a great challenge. The effect of these service conditions on the composite can range from a minor loss of stiffness at micro-level to catastrophic failures at structural level. The microstructure of the composite evolves in multiple ways before evidencing measurable degradation. Typical forms of micromechanical failures include fiber breaking, matrix cracking, fiber/matrix interface debonding etc. Factors such as microcracking (typically in a matrix) are unavoidable and can be inherent in the manufactured composite part. Microcracking can result from processing the composites at high temperatures (cure cycle), due to differences of thermal conductivities/coefficients of thermal expansion between the constituents (fiber, matrix). Other forms of composite material property degradation can result from hygrothermal loading and oxidation.

In this section, we will deal with the failure at ply level and not the micromechanical failures. Several failure theories have been developed to study failure envelopes of composite laminates. The failure theories that are available in FEMAP are discussed below. Some of these models (maximum stress theory, maximum strain theory) are based on pure comparison of observed stresses/strains in the laminate with their respective allowables. Other models such as Hill's theory, Tsai-Wu theory, and Hoffman's theory consider interaction of longitudinal/transverse stresses/strains to predict the failure envelope. Although failure theories can be handy to check the failure indices and decide if failure occurs in the laminate, it is important to understand the stress distribution in the model, interlaminar stresses and their effects on delamination, ABD matrices etc.

7.2 HILL'S THEORY

- Hill's failure theory is applicable for orthotropic materials that have the same strength in tension and compression, i.e., $X_t = X_c$ and $Y_t = Y_c$. Failure Index (FI) is given by:

$$\text{Failure Index} = \frac{\sigma_1^2}{X^2} - \frac{\sigma_1 \sigma_2}{X^2} + \frac{\sigma_2^2}{Y^2} + \frac{\tau_{12}^2}{S^2}$$

X is allowable stress in 1-direction

Y is allowable stress in 2-direction

S is allowable stress in shear

X_t = Allowable tensile stress in 1-direction

X_c = Allowable compressive stress in 1-direction

Y_t = Allowable tensile stress in 2-direction

Y_c = Allowable compressive stress in 2-direction

$X = X_t$ if σ_1 is positive or $X = X_c$ if σ_1 is negative and similarly for Y and σ_2 . For the interaction term $\sigma_1\sigma_2/X^2$, $X = X_t$ if σ_1 is positive or $X = X_c$ if σ_1 is negative.

Strength Ratio (SR) is given by,

$$SR = \frac{1}{\sqrt{FI}}$$

7.3 HOFFMAN'S THEORY

- Hoffman's theory for an orthotropic lamina in a general state of plane stress with unequal tensile and compressive strengths is given by,

$$\left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_1 + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_2 + \frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} + \frac{\sigma_{12}^2}{S^2} - \frac{\sigma_1 \sigma_2}{X_t X_c} = 1$$

- The failure index is obtained by evaluating the left-hand side of the above equation.

7.4 TSAI-WU THEORY

- The theory of strength for anisotropic materials proposed by Tsai and Wu specialized to the case of an orthotropic lamina in a general state of plane stress with unequal tensile and compressive strengths is,

$$F_1\sigma_1 + F_2\sigma_2 + F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + 2F_{12}\sigma_1\sigma_2 + F_{66}\sigma_{12}^2 = 1$$

$$F_1 = \frac{1}{X_t} - \frac{1}{X_c}$$

$$F_2 = \frac{1}{Y_t} - \frac{1}{Y_c}$$

$$F_{11} = +\frac{1}{X_t X_c}$$

$$F_{22} = +\frac{1}{Y_t Y_c}$$

$$F_{66} = +\frac{1}{S^2}$$

- The failure index is obtained by evaluating the left-hand side of the above equation.
- F_{12} is to be evaluated experimentally. By default, this term is set to zero in FEMAP (Tsai-Wu interaction term).

7.5 MAXIMUM STRAIN THEORY

- The maximum strain criterion has no strain interaction terms. The failure index is calculated using,

$$\text{Failure Index} = \text{MAX} \left[\left(\frac{\epsilon_1}{X} \right), \left(\frac{\epsilon_2}{Y} \right), \left(\frac{|\gamma_{12}|}{S} \right) \right]$$

- X , Y , and S are allowable strains in longitudinal direction, transverse direction and inplane, respectively.
- A failure index for maximum stress theory (available for 3D laminate modeling) can be derived similar to maximum strain theory.

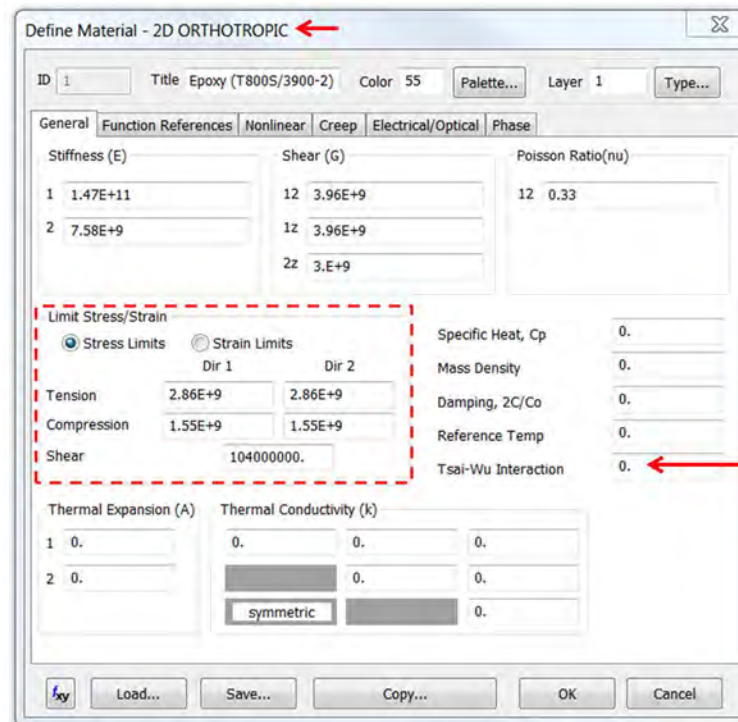
7.6 ONSET FAILURE THEORY

The Onset failure theory or the Strain Invariant Failure Theory (SIFT) is widely used in the aerospace industry. A brief overview of this failure criterion is provided in the appendix.

8. EXAMPLE 4: MODELING THE FAILURE BEHAVIOR OF COMPOSITE LAMINATES

For this example, we will use the 2D laminate model from example 1. All the modeling procedure that we have done for the 2D laminate model will be supplemented by defining failure strengths of the lamina and a failure criterion. The goal is to study the failure in composite laminates and also explore the output vectors that can be handy in visualizing the failure at ply level and laminate level.

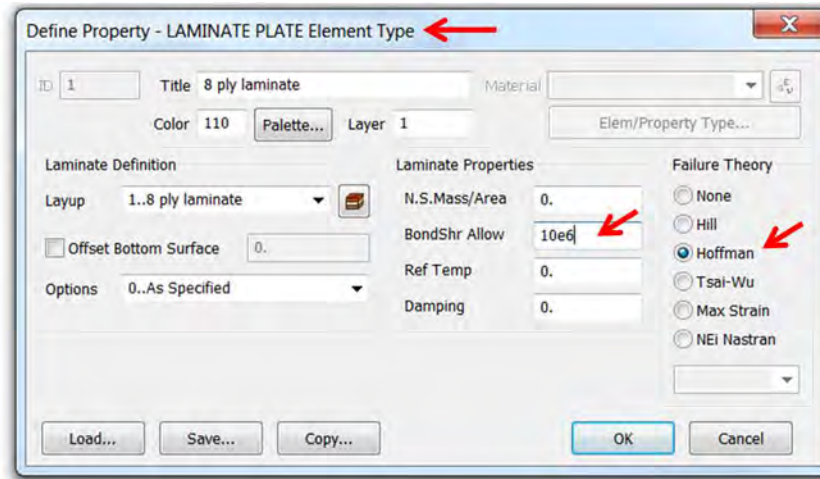
8.1 CREATING THE MATERIAL PROPERTY



Specify Tsai-Wu interaction term if you use Tsai-Wu failure criterion.

8.2 DEFINING THE LAMINATE PROPERTY

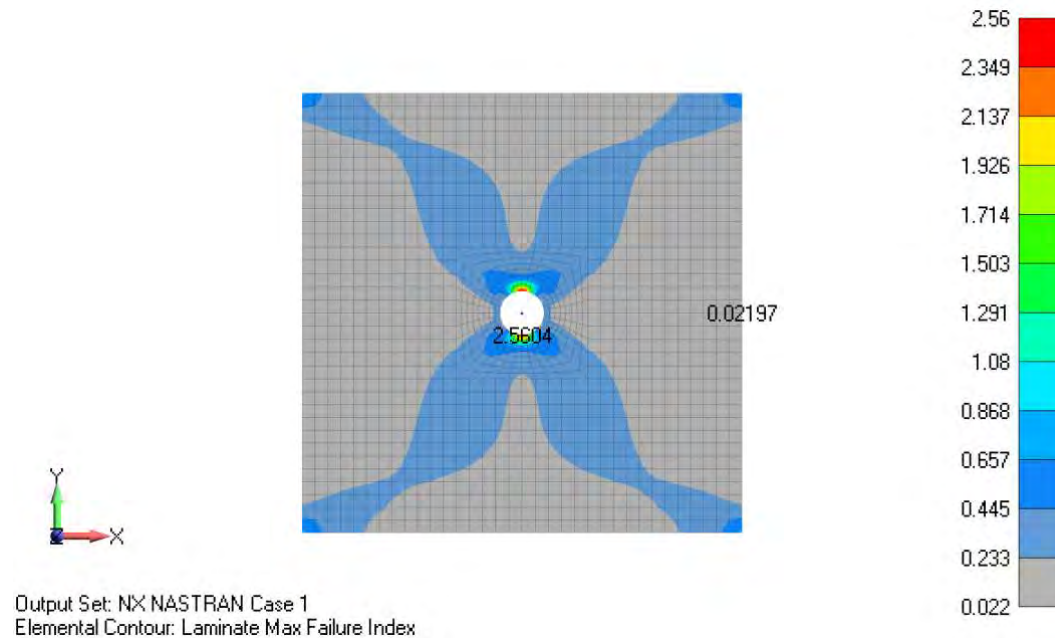
The laminate layup is the same as in example 1. The laminate property card is also defined in a similar manner. Additionally, we specify a failure criterion to calculate the failure indices at ply level and for the whole laminate. For this example, we will use the Hoffman's failure criterion. The bond shear allowable term is defined to predict interlaminar bond failure. If one is not interested in the interlaminar failure, this term can be left to its default value 0.



The specification of material angles follows the same approach as in example 1.

8.3 RESULTS

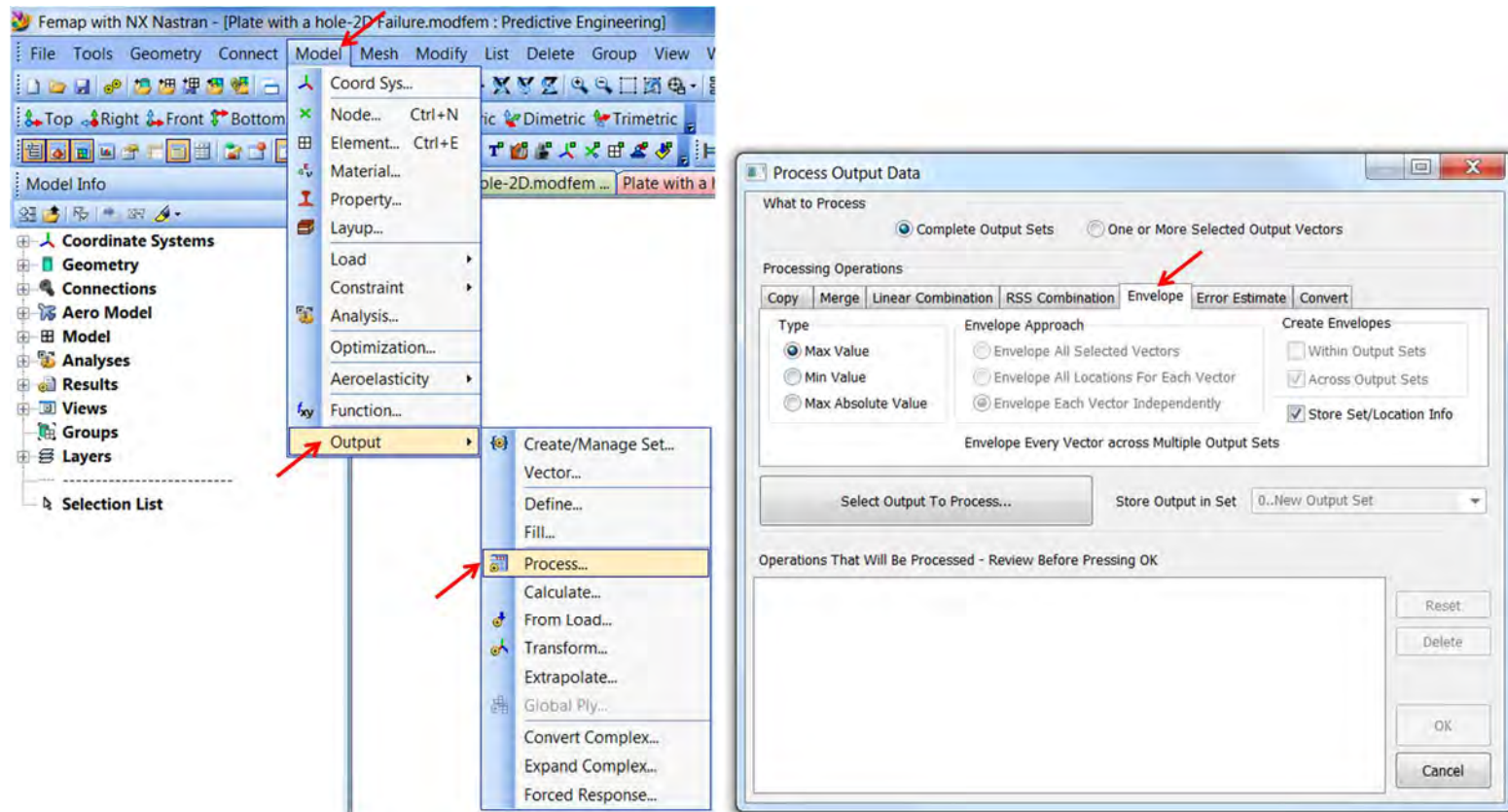
In this example, we are mainly interested at looking into the failure indices and evaluate the laminate and individual lamina. A uniaxial tensile load of 50 kN is applied. The contour below shows the laminate failure index and can be accessed from the output vector – 6060...Laminate Max Failure Index. This output vector shows the overall failure index of the laminate. A contour value (failure index) greater than or equal to 1 implies failure. Based on this information, one can assume that there are one or more layers in which the failure has occurred.



Next, we can check the failure index output vector on a ply-by-ply basis to find out the layers in which the failure has occurred. We can contour the output vector 'Lam Ply Fib Fail Index' for a particular ply and then use the custom tool options 'Laminate Next Ply' or 'Laminate Prev Ply' and check the failure indices.

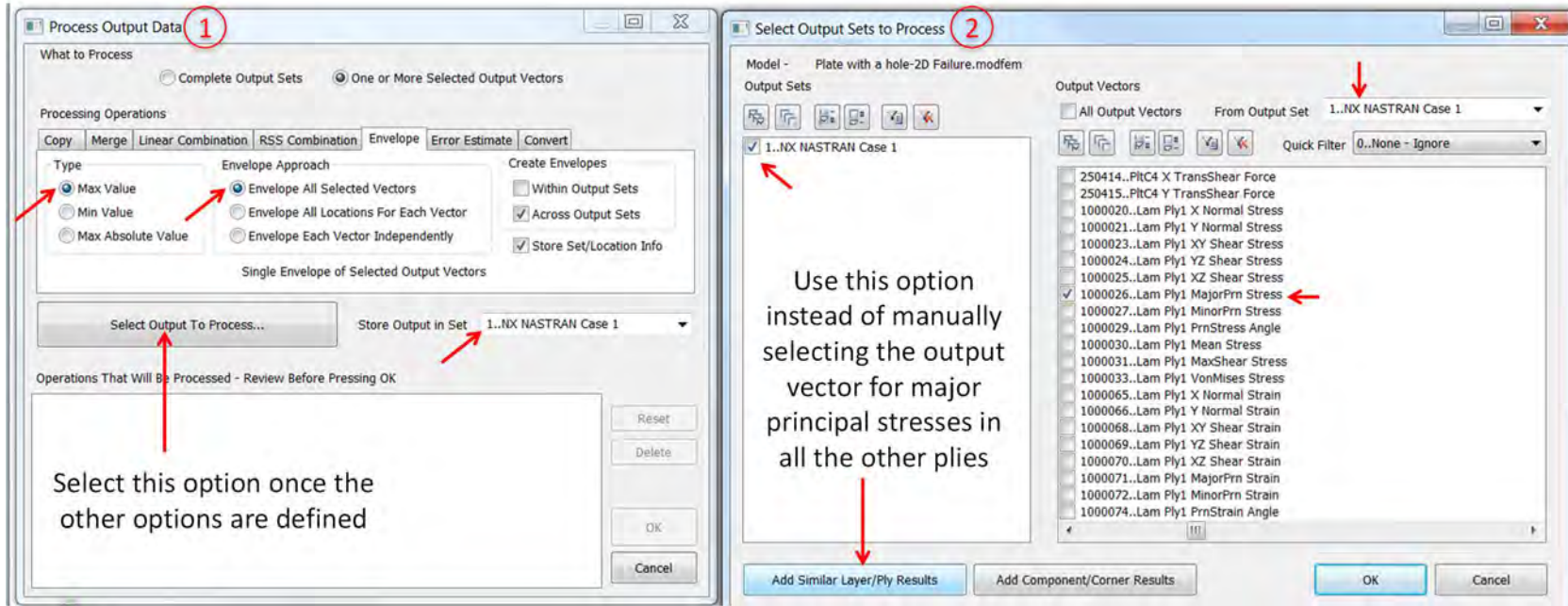
Within Custom Tools -> PostProcessing, FEMAP has an API for 'Laminate Envelope Failure Indices'. Currently, this API generates the same output vector as '6060...Laminate Max Failure Index'. However, this API can be custom modified to envelop other output vectors (e.g., bond failure indices).

Instead, if you are interested in manually selecting the output vectors and enveloping them to a single output vector, you can use Model -> Output -> Process



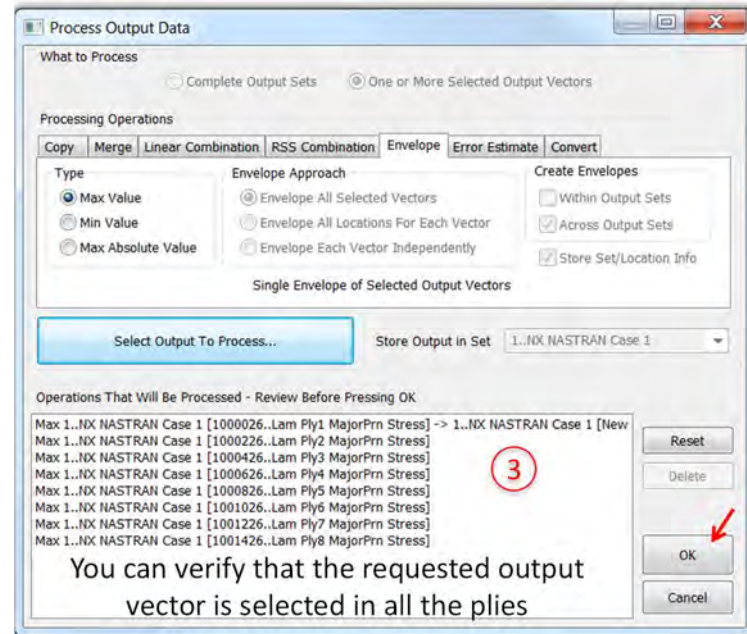
An example of enveloping major principal stresses in all the plies is shown below. This procedure can be followed to envelope any output vector.

If the analysis has only one output set, then we can envelope the output vectors from that output set. However, if we have multiple output sets, then we need to select one or more output sets from which we would like to envelope the output vectors.

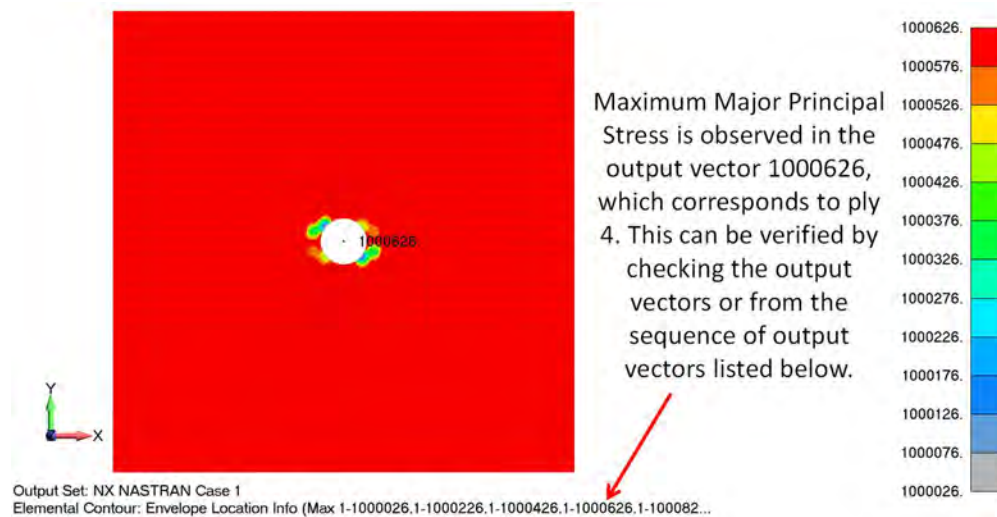


Select this option once the other options are defined

Use this option instead of manually selecting the output vector for major principal stresses in all the other plies



Other output vectors are also created which allow finding the location info (ply and element) of the maximum major principal stress. For example, from the above contour, we can see the location of the element with the maximum major principal stress. However, we do not know the respective ply that has this maximum value. The newly created output vector – ‘Envelope Location Info’ will have this information. One can modify the contour/criteria style to display only the max value, min value or both. Because, we are interested in max value only, required changes were done using F6 (View Options) -> PostProcessing -> Contour/Criteria Style -> Max Only. From the contour below, we can see that the maximum value of major principal stress is observed in ply 4.



9. ADDITIONAL READING

- Chapter 24: Laminates, NX Nastran User’s Guide
- ‘PCOMP and PCOMPS’ in NX Nastran Quick Reference Guide
- Chapter 6: Element Reference – ‘Laminate Element and Solid Laminate Element’ in FEMAP User Guide
- I. M. Daniel and O. Ishai, “Engineering Mechanics of Composite Materials,” 2nd Edition, 2005.
- R. M. Jones, “Mechanics of Composite Materials,” 2nd Edition, 1998.
- B. D. Agarwal, L. J. Broutman, and K. Chandrashekhara, “Analysis and Performance of Fiber Composites,” 3rd Edition, 2006.

10. MATERIAL DATABASE

This section provides material properties of some commonly used fibers (Table 2), polymer resins (

Table 3), and fiber-reinforced polymer composite materials (Table 4). Typically fibers are assumed to be isotropic. However, some references also considered transversely isotropic properties (different properties in longitudinal and transverse directions) for carbon fibers. Matrix materials are typically isotropic in nature.

Table 2: Mechanical properties of commonly used fibers [4]

Fiber	Tensile Modulus (GPa)	Tensile Strength (MPa)	Density (kg/m ³)
E-Glass	72.5	3500	2630
S-Glass	88	4600	2490
AS4 Carbon	245	4000	1800
IM7 Carbon	317	4900	1744
Kevlar 29	64	2860	1440
Kevlar 49	124	3650	1440
Boron	400	3620	2574

Table 3: Mechanical properties of commonly used polymer resins [5]

Resin	Tensile Modulus (GPa)	Tensile Strength (MPa)	Density (kg/m ³)
Epoxy	3.5	45	1200
Polycarbonate	2.7	77	1200
Polyethylene	0.7	33	950
Polyurethane	0.025	30	1200
Polyvinyl Chloride	1.5	60	1400

Table 4: Mechanical properties of commonly used fiber-reinforced polymer composite materials

Material	E_1 Msi (GPa)	E_2 Msi (GPa)	G_{12} Msi (GPa)	ν_{12}	V_f
T300/934 graphite/epoxy	19.0 (131)	1.5 (10.3)	1.0 (6.9)	0.22	0.65
AS/3501 graphite/epoxy	20.0 (138)	1.3 (9.0)	1.0 (6.9)	0.3	0.65
P-100/ERL 1962 graphite/epoxy	68.0 (468.9)	0.9 (6.2)	0.81 (5.58)	0.31	0.62
Kevlar 49/934 aramid/epoxy	11.0 (75.8)	0.8 (5.5)	0.33 (2.3)	0.34	0.65
E-glass/epoxy	5.6 (38.6)	1.2 (8.27)	0.6 (4.14)	0.26	0.45
Boron/5505 boron/epoxy	29.6 (204.0)	2.68 (18.5)	0.81 (5.59)	0.23	0.5

11. FOUR-POINT BENDING OF A SANDWICH COMPOSITE USING FEMAP AND NX NASTRAN

In this section, the simulation of four-point bending test (ASTM Standard D7249) of a sandwich composite is presented. A detailed hand calculation is presented to obtain the ply level stresses and strains. The simulation results are also compared with experimental data and hand calculation.

The geometry and loading configuration of the four-point bend specimen are shown below. Table 5 provides the applied load and the specimen geometry used in the four-point bend test.

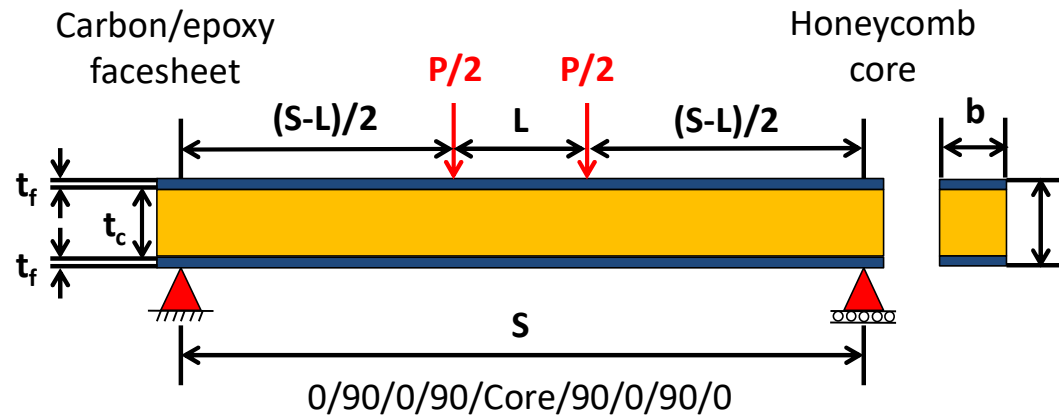


Table 5: Geometry and loading configuration of the four-point bend specimen

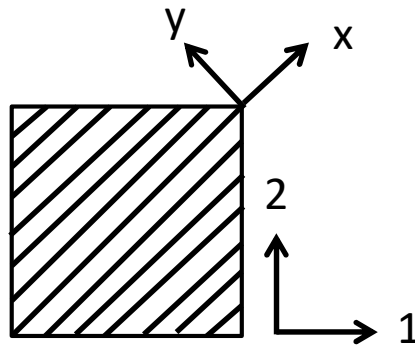
Parameter	Symbol	Value
Load	P	900.9 lbf
Support Span	S	22 in.
Load Span	L	4 in.
Beam Width	b	3 in.
Beam Thickness	t	1 in.
Facesheet Thickness	t_f	0.02 in.
Core Thickness	t_c	0.96 in.
Individual Ply Thickness	t_{ply}	0.005 in.

11.1 HAND CALCULATION

We will first show the hand calculations for a basic laminate which will provide a good foundation for sandwich composites. For sandwich composites, the concept and the equations are similar except that we have to account for facesheet offset from the midplane due to the core thickness.

11.1.1 BASIC LAMINATE

Consider an eight ply composite laminate with $[0/90/0/90]_s$ layup configuration ($n = 8$). We have an individual ply thickness $t_{ply} = 0.005$ in. We will use xyz -axis notation for material axis (ply level, local axis) and 123 -axis for laminate axis (global axis). This convention is consistent with the notation used in the textbook, "Introduction to Composite Materials," by Stephen W. Tsai and H. Thomas Hahn. This textbook was referred for the theory on flexural loading of composite laminates and sandwich composites.



We can calculate the stiffness terms Q_{ij} for each ply by using the following transformation.

	Q_{xx}	Q_{yy}	Q_{xy}	Q_{ss}
Q_{11}	m^4	n^4	$2m^2n^2$	$4m^2n^2$
Q_{22}	n^4	m^4	$2m^2n^2$	$4m^2n^2$
Q_{12}	m^2n^2	m^2n^2	$m^4 + n^4$	$-4m^2n^2$
Q_{66}	m^2n^2	m^2n^2	$-2m^2n^2$	$(m^2 - n^2)^2$
Q_{16}	m^3n	$-mn^3$	$mn^3 - m^3n$	$2(mn^3 - m^3n)$
Q_{26}	mn^3	$-m^3n$	$m^3n - mn^3$	$2(m^3n - mn^3)$
$m = \cos \theta, n = \sin \theta$				

To better understand the stiffness terms in material coordinate system, the following relations are provided. Here, σ_s corresponds to the inplane shear stress - σ_{xy} .

	ε_x	ε_y	ε_s
σ_x	Q_{xx}	Q_{xy}	
σ_y	Q_{xy}	Q_{yy}	
σ_s			Q_{ss}

The above relations are provided in terms of the compliance matrix as shown below ($Q_{ij} = S_{ij}^{-1}$).

	σ_x	σ_y	σ_s
ϵ_x	S_{xx}	S_{xy}	
ϵ_y	S_{xy}	S_{yy}	
ϵ_s			S_{ss}

It is easier to calculate the compliance matrix (S_{ij}) rather than the stiffness matrix (Q_{ij}). The compliance matrix terms can be calculated from the elastic properties of a unidirectional lamina ($E_x = 20.6 \text{ Msi}$, $E_y = 1.3 \text{ Msi}$, $E_s = G_{xy} = 0.55 \text{ Msi}$, $\nu_{xy} = 0.326$) as shown below.

$$S_{xx} = \frac{1}{E_x} = \frac{1}{20.6 \times 10^6} = 4.85 \times 10^{-8} \text{ (psi)}^{-1}$$

$$S_{yy} = \frac{1}{E_y} = \frac{1}{1.3 \times 10^6} = 7.69 \times 10^{-7} \text{ (psi)}^{-1}$$

$$S_{xy} = \frac{-\nu_{xy}}{E_x} = \frac{-0.326}{20.6 \times 10^6} = -1.58 \times 10^{-8} \text{ (psi)}^{-1}$$

$$S_{ss} = \frac{1}{E_s} = \frac{1}{0.55 \times 10^6} = 1.82 \times 10^{-6} \text{ (psi)}^{-1}$$

Now, the stiffness matrix can be easily calculated by inverting the compliance matrix.

$$\begin{bmatrix} Q_{xx} & Q_{xy} & 0 \\ Q_{xy} & Q_{yy} & 0 \\ 0 & 0 & Q_{ss} \end{bmatrix} = \begin{bmatrix} S_{xx} & S_{xy} & 0 \\ S_{xy} & S_{yy} & 0 \\ 0 & 0 & S_{ss} \end{bmatrix}^{-1}$$

We get,

$$Q_{xx} = 20.7 \times 10^6 \text{ psi}$$

$$Q_{yy} = 1.31 \times 10^6 \text{ psi}$$

$$Q_{xy} = 4.27 \times 10^5 \text{ psi}$$

$$Q_{ss} = 5.50 \times 10^5 \text{ psi}$$

Using the off-axis transformation relations (provided earlier), we can calculate the stiffness terms Q_{ij} $i, j = 1, 2, 6$.

Stiffness term	$\theta = 0$	$\theta = 90$
Q_{11}	$20.7 \times 10^6 \text{ psi}$	$1.31 \times 10^6 \text{ psi}$
Q_{22}	$1.31 \times 10^6 \text{ psi}$	$20.7 \times 10^6 \text{ psi}$
Q_{12}	$4.27 \times 10^5 \text{ psi}$	$4.27 \times 10^5 \text{ psi}$
Q_{66}	$5.50 \times 10^5 \text{ psi}$	$5.50 \times 10^5 \text{ psi}$
Q_{16}	0	0
Q_{26}	0	0

When a composite laminate is subjected to flexural loading, the moment-curvature relations are of interest rather than force-strain relations (inplane loading case). The moments and curvatures are related by an equivalent bending stiffness matrix for a multidirectional composite laminate. We can calculate the bending stiffness matrix D_{ij} $i, j = 1, 2, 6$. From the equation we can infer that D_{ij} is dependent on the stacking sequence.

$$D_{ij} = \frac{2}{3} t_{ply}^3 \sum_{k=1}^{n/2} Q_{ij}^k [k^3 - (k-1)^3]$$

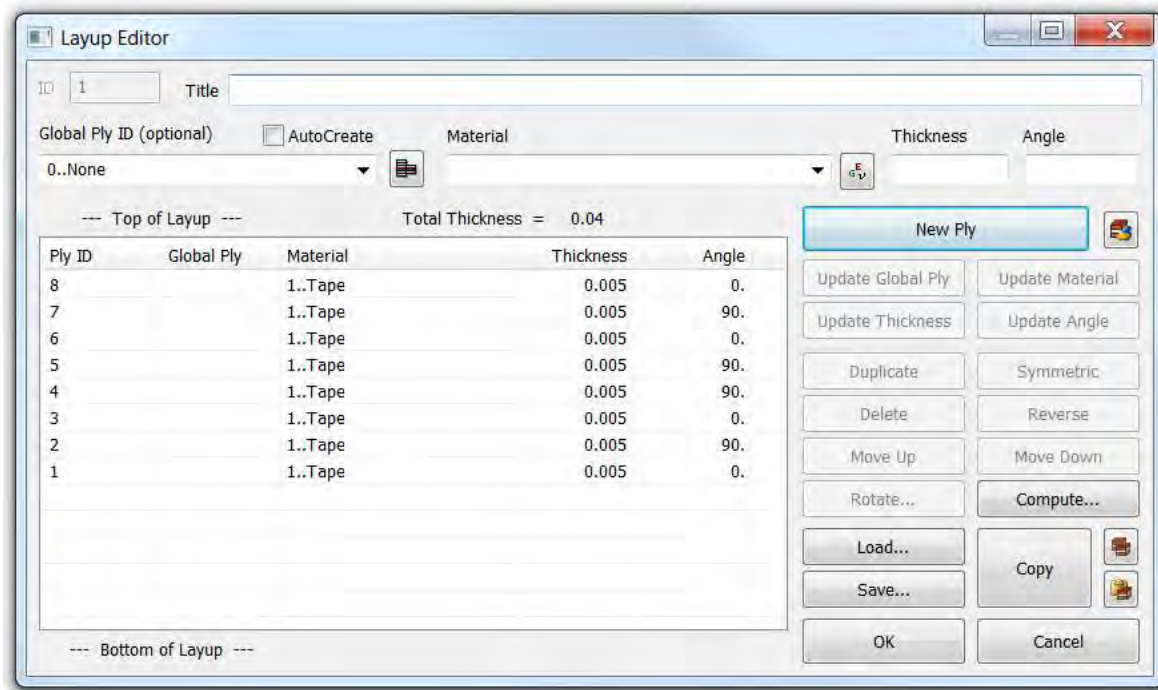
We get,

$$\begin{bmatrix} D_{11} & D_{12} & 0 \\ D_{12} & D_{22} & 0 \\ 0 & 0 & D_{66} \end{bmatrix} = \begin{bmatrix} 78.2 & 2.28 & 0 \\ 2.28 & 39.4 & 0 \\ 0 & 0 & 2.93 \end{bmatrix} \text{psi} \cdot \text{in}^3$$

We can calculate the bending compliance matrix by inverting the bending stiffness matrix $d_{ij} = D_{ij}^{-1}$.

$$\begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{12} & d_{22} & 0 \\ 0 & 0 & d_{66} \end{bmatrix} = \begin{bmatrix} 1.28 \times 10^{-2} & -7.40 \times 10^{-4} & 0 \\ -7.40 \times 10^{-4} & 2.54 \times 10^{-2} & 0 \\ 0 & 0 & 3.41 \times 10^{-1} \end{bmatrix} (\text{psi} \cdot \text{in}^3)^{-1}$$

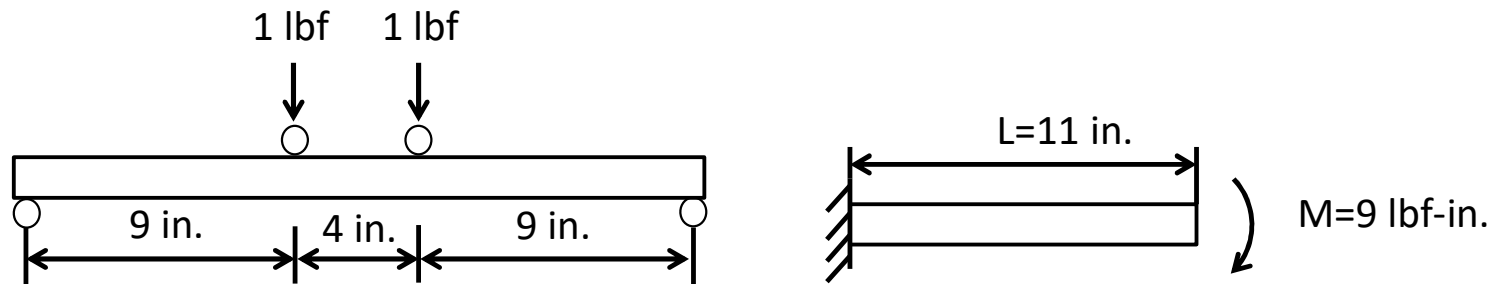
Both D_{ij} and d_{ij} matrices are compared with the values computed by FEMAP and they were matching quite well. One can always verify their hand calculations with the computed values from FEMAP to debug any errors. A snapshot of the layup configuration input to FEMAP is shown below.



The computed D_{ij} and d_{ij} matrices from FEMAP are compared with the hand calculations (shown below).

	Matrix Terms	Hand Calculation	FEMAP
Bending Stiffness Matrix D_{ij}	D_{11}	78.2	78.2
	D_{22}	39.4	39.4
	D_{12}	2.28	2.28
	D_{66}	2.93	2.94
Bending Compliance Matrix d_{ij}	d_{11}	1.28×10^{-2}	1.28×10^{-2}
	d_{22}	2.54×10^{-2}	2.54×10^{-2}
	d_{12}	-7.40×10^{-4}	-7.40×10^{-4}
	d_{66}	3.41×10^{-1}	3.40×10^{-1}

Next, we would like to analyze ply level stresses for the loading configuration shown below. The moment applied here is derived from a four-point bending load configuration with a total load of 2 lbf. From the bending moment diagram, the moment at half the length of the specimen is 9 lbf-in. The ply level stress evaluation in hand calculation does not account for the length of the specimen. However, the moment applied at the end will be used in calculating the stresses.



In order to calculate the ply level stresses and strains, we need to calculate the curvatures using the moments and d_{ij} . From curvatures, we can calculate the strains and then stresses from strains. The moment-curvature relation is shown below. Here the moment M_i is defined as the moment per unit width.

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_6 \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{21} & D_{22} & D_{26} \\ D_{61} & D_{62} & D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix}$$

The above relation can be expressed in terms of compliance as shown below ($[d_{ij}] = [D_{ij}]^{-1}$)

$$\begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & d_{16} \\ d_{21} & d_{22} & d_{26} \\ d_{61} & d_{62} & d_{66} \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_6 \end{Bmatrix}$$

In our example, we have $M_1 = -M/b$, $M_2 = 0$ and $M_6 = 0$. M is the total moment. We know the moments and we can calculate the curvatures as shown below.

$$\begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} = \begin{bmatrix} 1.28 \times 10^{-2} & -7.40 \times 10^{-4} & 0 \\ -7.40 \times 10^{-4} & 2.54 \times 10^{-2} & 0 \\ 0 & 0 & 3.41 \times 10^{-1} \end{bmatrix} \begin{Bmatrix} -M/b \\ 0 \\ 0 \end{Bmatrix}$$

The moment is $M = 9$ lbf-in. and the width of the specimen is $b = 3$ in.

$$\begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} = \begin{bmatrix} 1.28 \times 10^{-2} & -7.40 \times 10^{-4} & 0 \\ -7.40 \times 10^{-4} & 2.54 \times 10^{-2} & 0 \\ 0 & 0 & 3.41 \times 10^{-1} \end{bmatrix} \begin{Bmatrix} -3 \\ 0 \\ 0 \end{Bmatrix} = \begin{Bmatrix} -3.84 \times 10^{-2} \\ 2.22 \times 10^{-3} \\ 0 \end{Bmatrix}$$

We can calculate strains from curvatures.

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix} = z \begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix}$$

Here, z corresponds to the distance from the midplane of the laminate layup to midplane of a ply.

For the top most ply (0° ply), $z=0.0175$ in. The strains in the 0° ply are shown below.

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix} = 0.0175 \begin{Bmatrix} -3.84 \times 10^{-2} \\ 2.22 \times 10^{-3} \\ 0 \end{Bmatrix} = \begin{Bmatrix} -6.72 \times 10^{-4} \\ 3.89 \times 10^{-5} \\ 0 \end{Bmatrix}$$

We can calculate stresses in this ply using the following relation.

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_6 \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & Q_{16} \\ Q_{21} & Q_{22} & Q_{26} \\ Q_{61} & Q_{62} & Q_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix} = \begin{bmatrix} 20.7 \times 10^6 & 4.27 \times 10^5 & 0 \\ 4.27 \times 10^5 & 1.31 \times 10^6 & 0 \\ 0 & 0 & 5.50 \times 10^5 \end{bmatrix} \begin{Bmatrix} -6.72 \times 10^{-4} \\ 3.89 \times 10^{-5} \\ 0 \end{Bmatrix}$$

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_6 \end{Bmatrix} = \begin{Bmatrix} -13.9 \\ -0.24 \\ 0 \end{Bmatrix} \text{ ksi}$$

The stresses computed above are for the top ply and we see compressive stresses. Following the above procedure, we can calculate the stresses in all the other plies. The displacement for this loading configuration is calculated as shown below.

$$\delta = \frac{ML^2}{2EI} = \frac{ML^2}{2b} d_{11}$$

From hand calculations, the displacement was 2.3 in.

Table 6 provides the ply level stresses from hand calculations.

Table 6: Ply level stresses computed from the classical lamination theory for a $[0/90/0/90]_s$ layup

Ply Stresses (ksi)		Hand Calculation
Ply 1	σ_1	13.9
	σ_2	0.24
Ply 2	σ_1	0.62
	σ_2	-0.37
Ply 3	σ_1	5.97
	σ_2	0.10
Ply 4	σ_1	0.12
	σ_2	-0.07

11.1.2 SANDWICH COMPOSITE

In the previous section, all the required basics are discussed for analyzing the flexural behavior of sandwich composite. Here, we are adding a core and separating the laminated facesheets. We will show the hand calculations for a sandwich composite with layup configuration [0/90/0/90/Core/90/0/90/0]. The stiffness matrices (Q_{ij} $i, j = 1, 2, 6$) will remain the same as we are using the same orientations (calculated in the previous section for a [0/90/0/90]_s layup). Since the stacking sequence now includes a core, we have to account the core thickness (facesheet separation) in the bending stiffness matrix (D_{ij}) calculation.

When we have a core in the layup, it is treated as an equivalent number of plies. Accordingly, the limits on the summation in the bending stiffness matrix [D_{ij}] are modified as shown below.

$$D_{ij} = \frac{2}{3} t_{ply}^3 \sum_{k=n_c/2+1}^{n/2} Q_{ij}^k [k^3 - (k-1)^3]$$

Here, n is the total number of equivalent plies in the sandwich composite and n_c is the number of equivalent plies in a core. In our example, we have a total thickness of 1 in. for the sandwich composite and 0.96 in. for the core. We have an individual ply thickness $t_{ply} = 0.005$ in. So, $n=1/0.005=200$ and $n_c=0.96/0.005=192$. In the above equation, k is the ply number and ranges from 97 to 100. We can also see that the bending stiffness matrix accounts for the stiffness of the facesheet only.

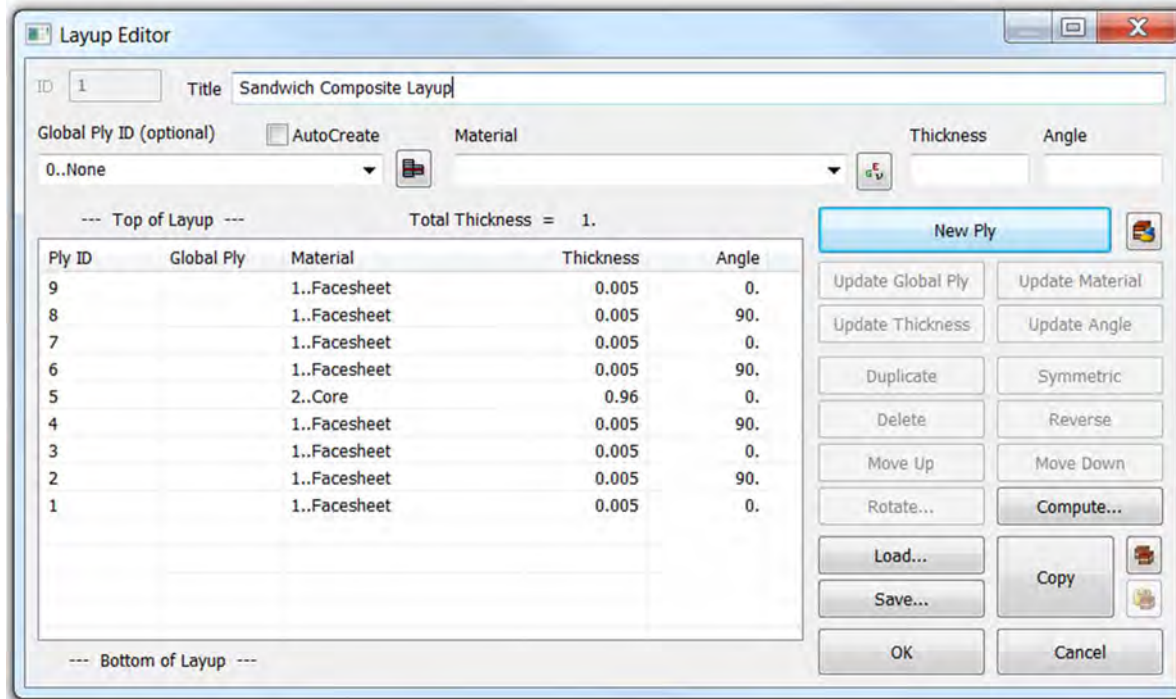
Substituting Q_{ij} of 0° and 90° plies from the previous section, we get,

$$\begin{bmatrix} D_{11} & D_{12} & 0 \\ D_{12} & D_{22} & 0 \\ 0 & 0 & D_{66} \end{bmatrix} = \begin{bmatrix} 1.07 \times 10^5 & 4.10 \times 10^3 & 0 \\ 4.10 \times 10^3 & 1.05 \times 10^5 & 0 \\ 0 & 0 & 5.28 \times 10^3 \end{bmatrix} \text{psi} \cdot \text{in}^3$$

We can calculate the bending compliance matrix by inverting the bending stiffness matrix $d_{ij} = D_{ij}^{-1}$.

$$\begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{12} & d_{22} & 0 \\ 0 & 0 & d_{66} \end{bmatrix} = \begin{bmatrix} 9.37 \times 10^{-6} & -3.66 \times 10^{-7} & 0 \\ -3.66 \times 10^{-7} & 9.54 \times 10^{-6} & 0 \\ 0 & 0 & 1.89 \times 10^{-4} \end{bmatrix} (\text{psi} \cdot \text{in}^3)^{-1}$$

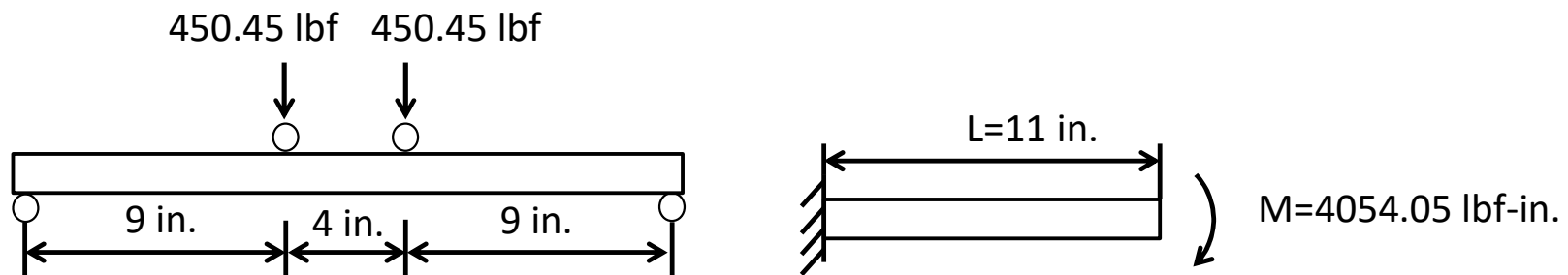
Similar to the previous section, we would like to verify the hand calculations with the FEMAP computed values. The layup configuration input to FEMAP is shown below.



The computed D_{ij} and d_{ij} matrices from FEMAP are compared with the hand calculations (shown below).

	Matrix Terms	Hand Calculation	FEMAP
Bending Stiffness Matrix D_{ij}	D_{11}	1.07×10^5	1.07×10^5
	D_{22}	1.05×10^5	1.05×10^5
	D_{12}	4.10×10^3	4.10×10^3
	D_{66}	5.28×10^3	5.30×10^3
Bending Compliance Matrix d_{ij}	d_{11}	9.37×10^{-6}	9.37×10^{-6}
	d_{22}	9.54×10^{-6}	9.54×10^{-6}
	d_{12}	-3.66×10^{-7}	-3.66×10^{-7}
	d_{66}	1.89×10^{-4}	1.89×10^{-4}

Next, we would like to analyze ply level stresses for the loading configuration shown below. The moment applied here is derived from a four-point bending load configuration with a total load of 900.9 lbf. From the bending moment diagram, the moment at half the length of the specimen is 4054 lbf-in.



The moment-curvature relation is shown below. Here the moment M_i is defined as the moment per unit width.

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_6 \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{21} & D_{22} & D_{26} \\ D_{61} & D_{62} & D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix}$$

The above relation can be expressed in terms of compliance as shown below ($[d_{ij}] = [D_{ij}]^{-1}$)

$$\begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & d_{16} \\ d_{21} & d_{22} & d_{26} \\ d_{61} & d_{62} & d_{66} \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_6 \end{Bmatrix}$$

In our example, we have $M_1 = -M/b$, $M_2 = 0$ and $M_6 = 0$. M is the total moment. We know the moments and we can calculate the curvatures as shown below.

$$\begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} = \begin{bmatrix} 9.37 \times 10^{-6} & -3.66 \times 10^{-7} & 0 \\ -3.66 \times 10^{-7} & 9.54 \times 10^{-6} & 0 \\ 0 & 0 & 1.89 \times 10^{-4} \end{bmatrix} \begin{Bmatrix} -M/b \\ 0 \\ 0 \end{Bmatrix}$$

The moment is $M = 4054$ lbf-in. and the width of the specimen is $b = 3$ in.

$$\begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} = \begin{bmatrix} 9.37 \times 10^{-6} & -3.66 \times 10^{-7} & 0 \\ -3.66 \times 10^{-7} & 9.54 \times 10^{-6} & 0 \\ 0 & 0 & 1.89 \times 10^{-4} \end{bmatrix} \begin{Bmatrix} -1351.35 \\ 0 \\ 0 \end{Bmatrix} = \begin{Bmatrix} -1.27 \times 10^{-2} \\ 4.95 \times 10^{-4} \\ 0 \end{Bmatrix}$$

We can calculate strains from curvatures.

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix} = z \begin{Bmatrix} \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix}$$

Here, z corresponds to the distance from the midplane of the sandwich composite to midplane of the corresponding ply layer.

For the top most ply (0° ply), $z=0.4975$ in. The strains in the 0° ply are shown below.

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix} = 0.4975 \begin{Bmatrix} -1.27 \times 10^{-2} \\ 4.95 \times 10^{-4} \\ 0 \end{Bmatrix} = \begin{Bmatrix} -6.30 \times 10^{-3} \\ 2.46 \times 10^{-4} \\ 0 \end{Bmatrix}$$

We can calculate stresses in this ply using the following relation.

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_6 \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & Q_{16} \\ Q_{21} & Q_{22} & Q_{26} \\ Q_{61} & Q_{62} & Q_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix} = \begin{bmatrix} 20.7 \times 10^6 & 4.27 \times 10^5 & 0 \\ 4.27 \times 10^5 & 1.31 \times 10^6 & 0 \\ 0 & 0 & 5.50 \times 10^5 \end{bmatrix} \begin{Bmatrix} -6.30 \times 10^{-3} \\ 2.46 \times 10^{-4} \\ 0 \end{Bmatrix}$$

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_6 \end{Bmatrix} = \begin{Bmatrix} -131 \\ -2.37 \\ 0 \end{Bmatrix} \text{ ksi}$$

Since it is the top most ply, the stresses are compressive. Similarly, we can calculate the stresses in other plies by changing the distance z and also accounting for off-axis to on-axis transformation.

Table 7 provides the ply level stresses from hand calculations.

Table 7: Ply level stresses computed from the classical lamination theory for a [0/90/0/90/Core/90/0/90/0] layup

Ply Stresses (ksi)		Hand Calculation
Ply 1	σ_1	131
	σ_2	2.37
Ply 2	σ_1	8.06
	σ_2	-2.39
Ply 3	σ_1	128
	σ_2	2.32
Ply 4	σ_1	7.90
	σ_2	-2.34

For the four-point bending problem, the deflection $\delta = \frac{P\left(\frac{S-L}{2}\right)(3S^2 - (S-L)^2)}{48EI}$. Here, EI can be expressed as $\frac{b}{d_{11}}$ (refer to the textbook)

$$\delta = \frac{P\left(\frac{S-L}{2}\right)(3S^2 - (S-L)^2)}{48b} d_{11} = 0.6 \text{ in.}$$

11.2 FINITE ELEMENT SIMULATION

A half-symmetric model of the four-point bending specimen was used in the finite element simulation. It is important to note this half symmetry approach will not be applicable when we have ply orientation angles other than 0° and 90°. The sandwich composite is modeled using laminate plate elements (procedure discussed in section 6.6 – method 1). The screen shots of the material properties used for the facesheet and the core are provided below. Imperial unit system was used for this example. The classical lamination theory assumes the laminated plate is non-shear deformable. Accordingly, the transverse shear moduli of the core are set to a high value.

Define Material - 2D ORTHOTROPIC

ID 1 Title **Facesheet** Color 55 Palette... Layer 1 Type...

General Function References Nonlinear Creep Electrical/Optical Phase

Stiffness (E)	Shear (G)	Poisson Ratio(nu)
1 20600000.	12 551631.	12 0.326
2 1300000.	1z 551631.	
	2z 370000.	

Limit Stress/Strain
 Stress Limits Strain Limits

Tension Dir 1 0. Dir 2 0.
 Compression Dir 1 0. Dir 2 0.
 Shear 0.

Specific Heat, Cp 0.
 Mass Density 0.
 Damping, 2C/Co 0.
 Reference Temp 0.
 Tsai-Wu Interaction 0.

Thermal Expansion (A)
 1 0.
 2 0.

Thermal Conductivity (k)
 0. 0. 0.
 0. 0. 0.
 symmetric 0.

fy Load... Save... Copy... OK Cancel

Define Material - 2D ORTHOTROPIC

ID 2 Title **Core** Color 55 Palette... Layer 1 Type...

General Function References Nonlinear Creep Electrical/Optical Phase

Stiffness (E)	Shear (G)	Poisson Ratio(nu)
1 0.01	12 0.01	12 1.E-5
2 0.01	1z 100000000.	
	2z 100000000.	

Limit Stress/Strain
 Stress Limits Strain Limits

Tension Dir 1 0. Dir 2 0.
 Compression Dir 1 0. Dir 2 0.
 Shear 0.

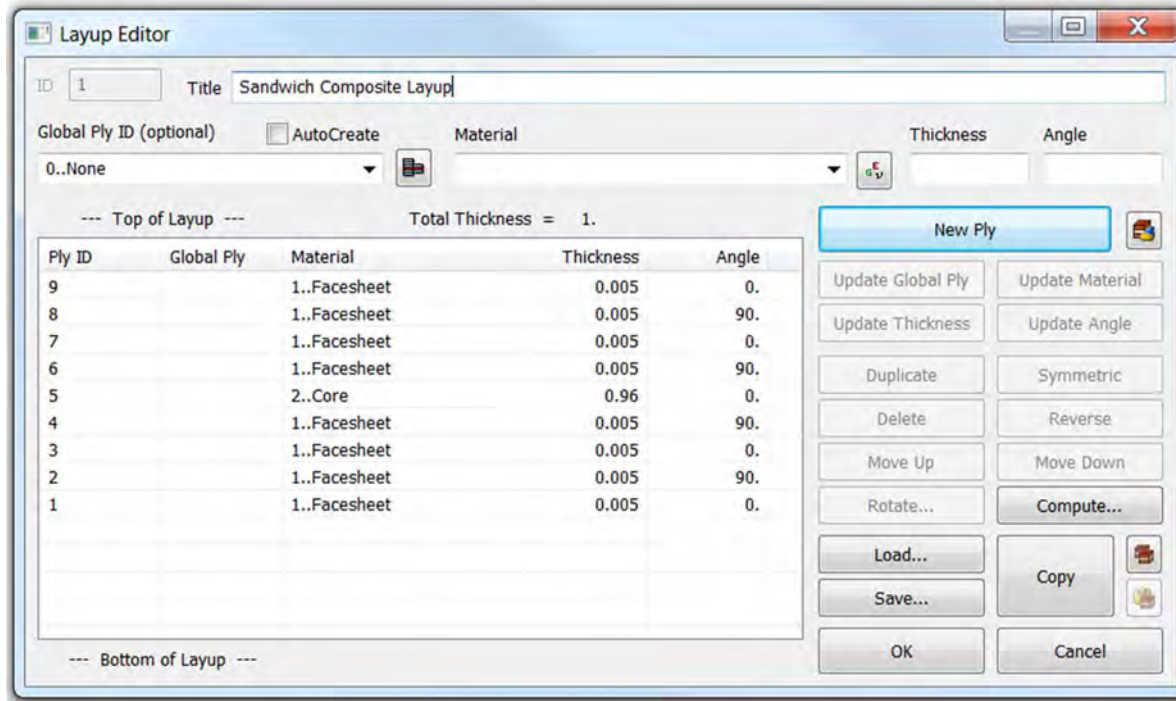
Specific Heat, Cp 0.
 Mass Density 0.
 Damping, 2C/Co 0.
 Reference Temp 0.
 Tsai-Wu Interaction 0.

Thermal Expansion (A)
 1 0.
 2 0.

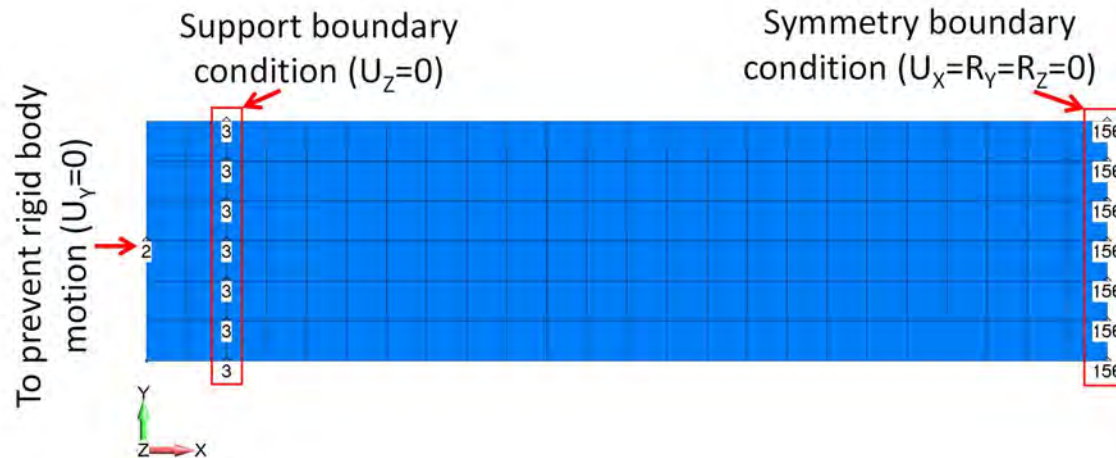
Thermal Conductivity (k)
 0. 0. 0.
 0. 0. 0.
 symmetric 0.

fy Load... Save... Copy... OK Cancel

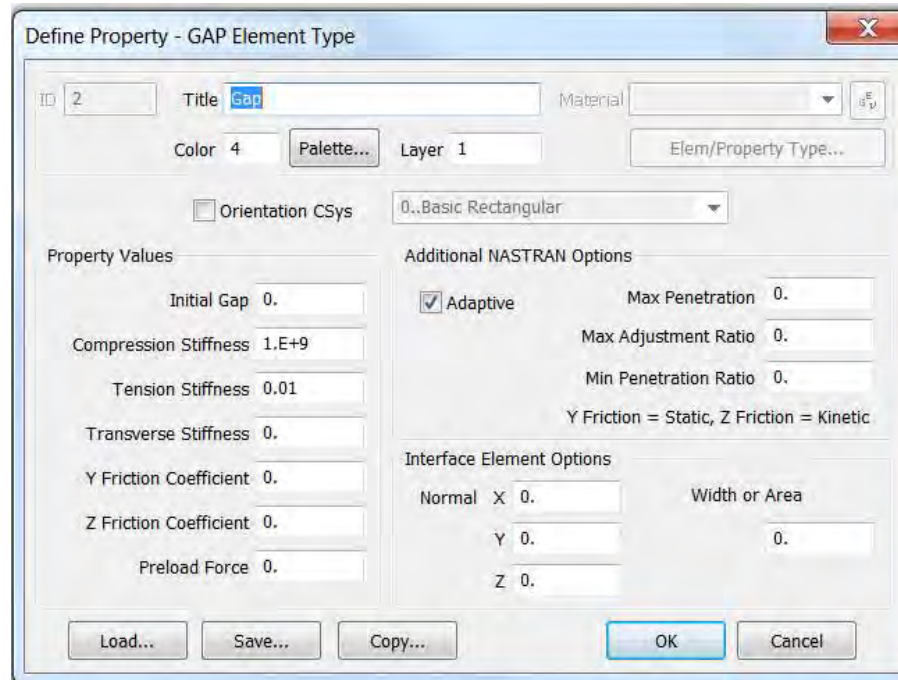
A screen shot of the layup configuration used in the finite element model is shown below.



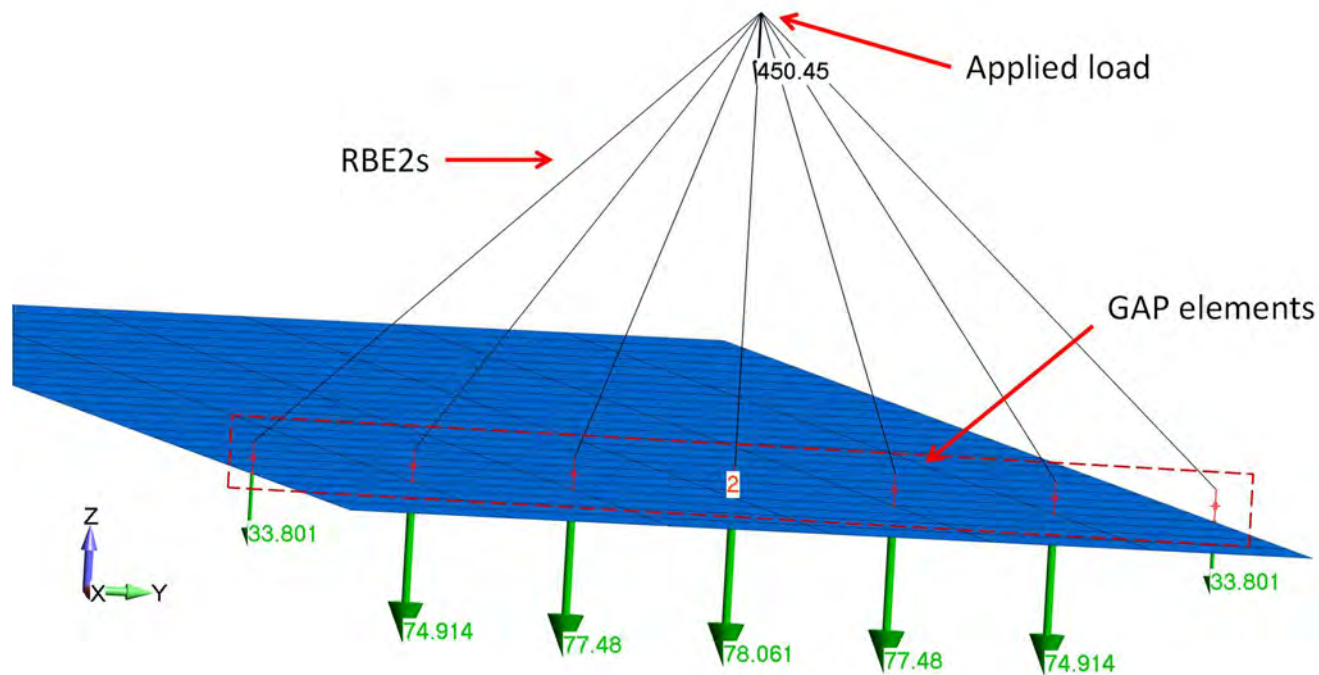
The boundary conditions used in the finite element model are shown below.



The required load ($P/2=450.45$ lbf) was applied on the nodes using a combination of GAP elements and RBE2s as shown below. This procedure allows for a better distribution of loads on the nodes. A new set of nodes have been created by duplicating and offsetting the nodes along the loading region by a finite distance (here 0.1 in.). These new set of nodes are connected to the corresponding nodes on the loading region using GAP elements. The GAP element properties (i.e., compression stiffness is adjusted in an increasing fashion until there is a minimal change ($\sim 0.01\%$) in the forces distributed on the nodes. A snapshot of the GAP element properties is shown below.

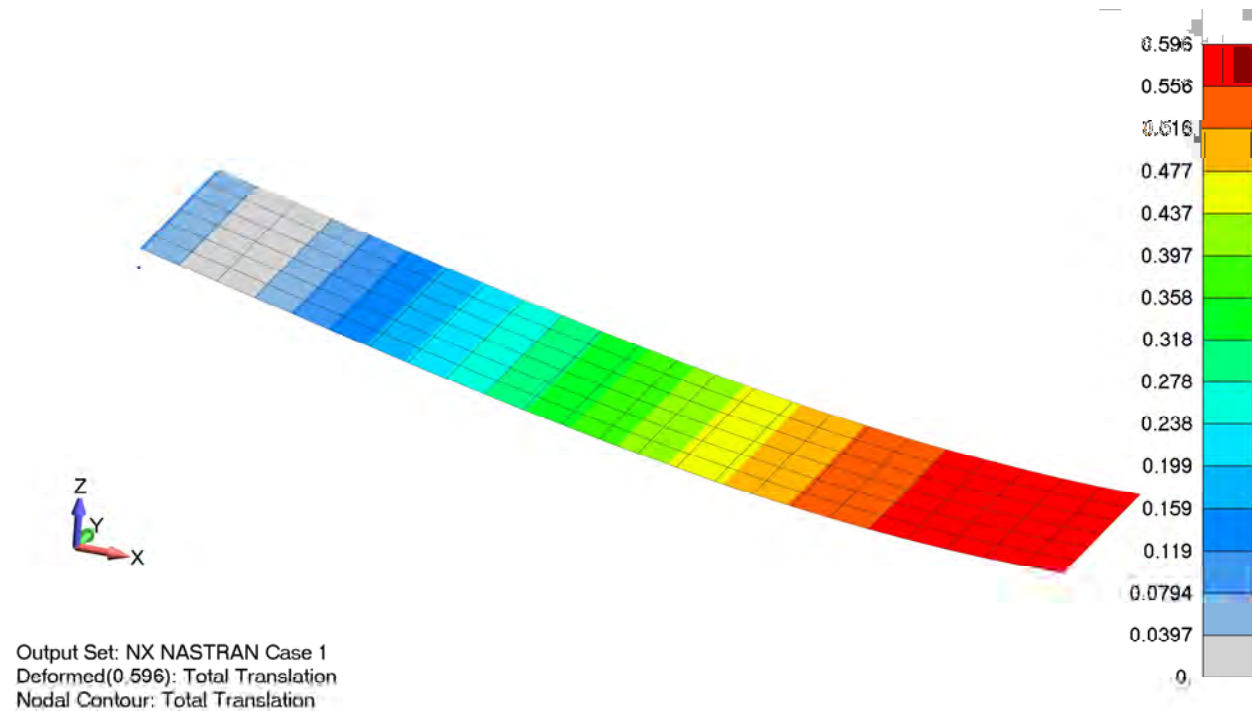


The load application procedure is illustrated below. The freebody diagram shows the nodal force distribution. One can manually calculate the force at each node (accounting for difference in the contribution of force at end nodes and mid nodes) and compare them with the values obtained using the GAP elements. The latter procedure will give a more gradual distribution of forces as compared to the former case. A nonlinear static analysis is used in this case (with GAP elements). This procedure is used as an initial step to get the load distribution at each node. GAP elements are nonlinear elements and will involve stiffness updates during the analysis depending on the GAP status (i.e., open or closed). These GAP stiffness updates will modify the global stiffness matrix.

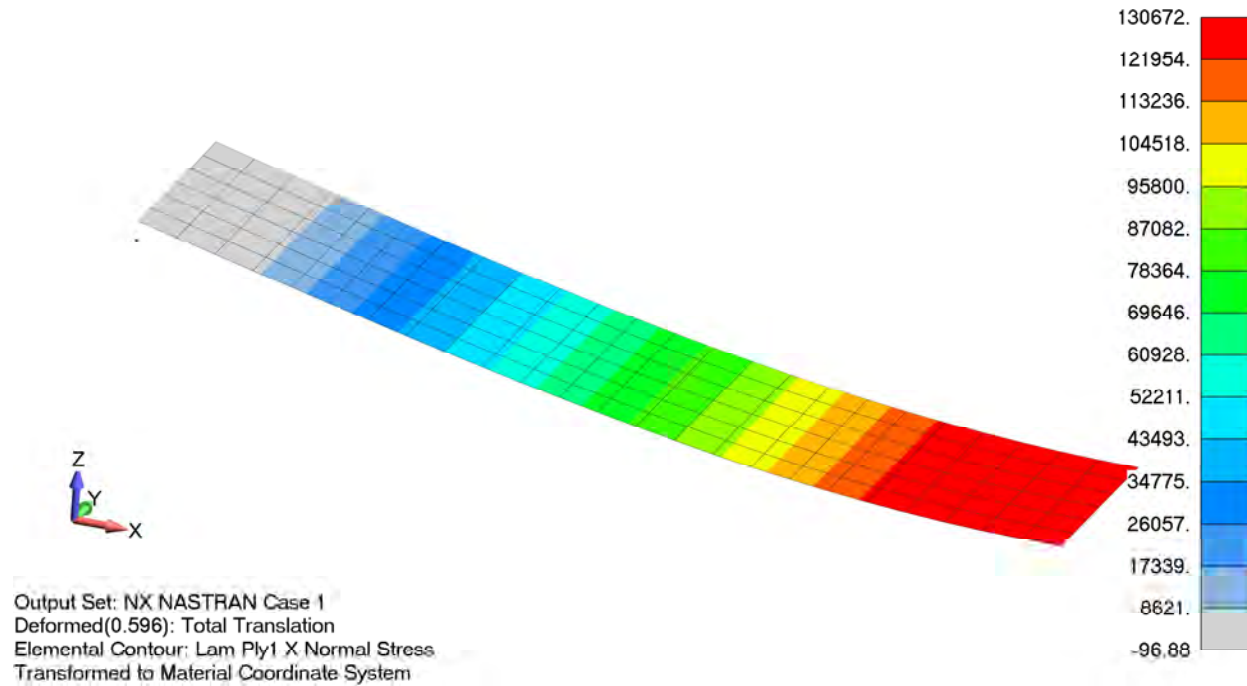


In this example GAP elements are used only to get the load distribution on the nodes. A linear static analysis is performed with the nodal forces obtained from a nonlinear static analysis. In this case, no GAP elements are used and the forces are directly applied on the nodes.

From the finite element simulation, the maximum displacement in the sandwich composite under four-point bending is 0.6 in.



The maximum tensile stress in the bottom ply (0° - ply 1) is 131 ksi.



An important note on post processing the simulation results for off-axis plies (other than 0° - for example 90°, 45° etc.) is that NX Nastran prints out stresses in the fiber and material directions and not the element coordinate system (as with an isotropic material). We can check this by printing the stresses in the .fo6 file.

A snapshot of the stresses printed in the .fo6 file is shown below.

Typically, for an isotropic material, would print out stresses in the element coordinate system and uses Normal-X and Normal-Y instead of Normal-1 and Normal-2

```

139      7 -1.22422E+05  2.91883E+03 -3.44956E+01  3.80329E+00 -1.90247E+00 -89.99  2.91884E+03 -1.22422E+05  6.26706E+04
139      8 -1.98795E+04 -1.55889E+03  6.66818E+03  2.92646E+00 -3.30989E-01  74.13  5.29678E+01 -2.14914E+04  1.07722E+04
1 NX NASTRAN NL STATIC ANALYSIS SET                                JANUARY 22 2015 NX NASTRAN 5/1/14 PAGE 59
0
LOAD STEP = 1 00000E+00
          STRESSES IN FIBER AND MATRIX DIRECTIONS
ELEMENT ID  NORMAL-1  NORMAL-2  SHEAR-12  SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE  MAJOR  MINOR  MAX SHEAR
139      9 -1.24920E+05  2.97819E+03 -3.10794E+01  -4.12744E-16  2.31167E-16  -89.99  2.97819E+03 -1.24920E+05  6.39489E+04
140      1  1.23915E+05 -3.00363E+03 -7.63881E-01  8.16920E-01 -2.23307E-01  -0.00  1.23915E+05 -3.00363E+03  6.34595E+04
140      2  1.87632E+01  1.53038E+03 -5.6105E-01  1.06169E+00 -1.28353E+00  -16.73  3.01383E+01 -1.45591E+02  1.02919E+04
140      3  1.21410E+05 -2.94265E+03 -7.54623E-01  1.86218E+00 -1.50235E+00  -0.00  1.21410E+05 -2.94265E+03  6.21764E+04
140      4  1.83998E+04  1.50692E+03  5.52768E+03  2.10198E+00 -2.54104E+00  16.60  2.00478E+04 -1.41080E+02  1.00944E+04
140      5 -3.55557E-07  3.47460E-07 -5.49796E-09  2.10198E+00 -2.54104E+00  -89.55  3.47503E-07 -3.55600E-07  3.53551E-07
140      6 -1.84057E+04 -1.50881E+03 -5.60524E+03  1.86218E+00 -1.50235E+00  -73.22  1.81530E+02 -2.00961E+04  1.01388E+04
140      7 -1.22855E+05  3.00326E+03  1.48053E-01  1.06169E+00 -1.28353E+00  90.00  3.00326E+03 -1.22855E+05  6.29293E+04
140      8 -1.87815E+04 -1.54030E+03  5.72061E+03  8.16920E-01 -2.23307E-01  73.22  1.85116E+02 -2.05069E+04  1.03460E+04
140      9 -1.25361E+05  3.06424E+03  1.57312E-01 -1.15217E-16  1.55960E-16  90.00  3.06424E+03 -1.25361E+05  6.42124E+04
141      1  1.23913E+05 -3.00250E+03 -2.25881E+01 -8.93088E-01 -2.21659E-01  -0.01  1.23913E+05 -3.00251E+03  6.34579E+04
141      2  1.83638E+04  1.55732E+03 -5.64249E+03 -1.16068E+00 -1.27406E+00  -16.94  2.00824E+04 -1.61306E+02  1.01218E+04
141      3  1.21408E+05 -2.94154E+03 -2.21260E+01 -2.03581E+00 -1.49126E+00  -0.01  1.21408E+05 -2.94154E+03  6.21748E+04
141      4  1.87971E+04  1.49067E+03  5.52713E+03 -2.29797E+00 -2.52229E+00  16.28  2.04117E+04 -1.23899E+02  1.02678E+04
141      5 -3.55594E-07  3.47542E-07  7.24465E-09 -2.29797E+00 -2.52229E+00  89.41  3.47617E-07 -3.55669E-07  3.51643E-07
141      6 -1.88288E+04 -1.49143E+03 -5.60471E+03 -2.03581E+00 -1.49126E+00  -73.56  1.62619E+02 -2.04829E+04  1.03228E+04
141      7 -1.22853E+05  3.00217E+03  2.29253E+01 -1.16068E+00 -1.27406E+00  89.99  3.00217E+03 -1.22853E+05  6.29277E+04
141      8 -1.83661E+04 -1.55936E+03  5.72006E+03 -8.93088E-01 -2.21659E-01  72.88  2.02693E+02 -2.01281E+04  1.01654E+04
141      9 -1.25359E+05  3.06313E+03  2.33874E+01  1.25960E-16  1.54809E-16  89.99  3.06313E+03 -1.25359E+05  6.42109E+04
    
```

Notice that Normal-1 and Normal-2 convention is used unlike Normal-X and Normal-Y. However, while post processing FEMAP uses these stresses and stores them in the output vector – Lam Ply X Normal Stress and Lam Ply Y Normal Stress. Although FEMAP calls it X Normal Stress and Y Normal Stress, they correspond to Normal-1 and Normal-2 respectively. This implies that Lam Ply X Normal Stress is in the ply fiber direction and Lam Ply Y Normal Stress is in the ply matrix direction (transverse direction). If one is comparing with the hand calculations, these stresses have to be transformed as hand calculations correspond to global X- and Y- directions (or 0° angle for the entire laminate as specified in the property card). In this example the material angle (specified on element property card) is in the global X-direction (shown below).



Since ply 1 is 0° (in this example), no transformation is required. Ply 2 is 90° and requires transformation. Accordingly, the stresses can be transformed to global –X and –Y as shown below. These transformations have to be used carefully as they would significantly change the results. Another option is to transform the stresses in the hand calculations to the ply orientation angle (no transformation in FEMAP is required in that case).

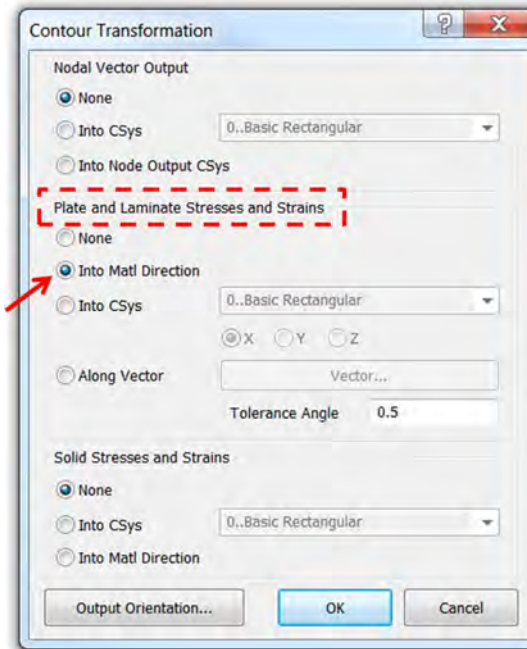


Plate and Laminate Stresses and Strains

The options for Plate Forces, Stresses and Strains and Laminate/Solid Laminate Stresses and Strains are:

- *None* - no transformation (default)
- *Into Matl Direction* - transforms output using the predefined "material angle" specified for each element. You can set the "material angle" when creating plane properties (in *Define Property* dialog box, choose *Elem/Property Type*, then click *Element Material Orientation*) or at anytime using the *Modify, Update Elements, Material Angle* command (for more information, see [Section 4.8.3.13, "Modify, Update Elements, Material Orientation..."](#)), which has several options.
- *Into CSys* - transforms the output vector to align the X-direction of output vector to the chosen *X*, *Y*, or *Z* component of an existing coordinate system.
- *Along Vector* - transforms the output vector to align the X-direction of output vector to a vector specified by clicking the *Vector* button, then using the standard vector definition dialog box.

11.3 SUMMARY OF RESULTS

From the four-point bending experimental testing, the maximum deflection for the applied load was 0.85 in. From the finite element simulation and hand calculations, the calculated displacements for the [0/90/0/90/Core/90/0/90/0] layup were 0.6 in. each. It should be noted that the thickness of each ply used in hand calculations and finite element simulation was 0.005 in. Accordingly, the sandwich into an equivalent number of plies giving a round number for the hand calculation. If an actual thickness from the experiment was used, a deflection within 10% difference from the experimental value could be obtained.

The stresses calculated using the finite element simulation and hand calculations are shown below. The finite element simulation was repeated for [0/45/0/-45/Core/-45/0/45/0] using a full model (no symmetry due to 45° plies) and the stresses are compared with hand calculations. It should be noted that the classical lamination theory does not account for ply level stress distribution in the specimen width direction. When a [0/90/0/90/Core/90/0/90/0] was used, the stresses are uniform in the width direction. However, for a [0/45/0/-45/Core/-45/0/45/0] layup, the stresses vary in the width direction due to the off-axis plies. As a result, the simulated stresses deviate from the hand calculations when actual specimen widths are used in the finite element model. To reduce this effect, an infinitesimally thin width specimen has to be modeled when off-axis plies are used such that the model conforms to the classical lamination theory assumptions and so does the ply level stresses.

Table 8 provides the comparison of the ply level stresses computed using hand calculations and finite element simulations.

Table 8: Comparison of ply level stresses between hand calculations and FEA results

Ply		0/90/0/90/Core/90/0/90/0			0/45/0/-45/Core/-45/0/45/0		
		Finite Element Analysis	Hand Calculation	Difference (%)	Finite Element Analysis**	Hand Calculation	Difference (%)
Ply 1	σ_1 (ksi)	131	131	0	124	124	0
	σ_2 (ksi)	2.37	2.38	0.4	-3.26	-3.26	0
Ply 2	σ_1 (ksi)	-2.39	-2.40*	0.4	16.5	16.5*	0
	σ_2 (ksi)	8.06	8.10*	0.5	1.38	1.39*	0.7
Ply 3	σ_1 (ksi)	128	129	0.8	121	121	0
	σ_2 (ksi)	2.32	2.33	0.4	-3.20	-3.20	0
Ply 4	σ_1 (ksi)	-2.34	-2.36*	0.8	16.6	16.7*	0.6
	σ_2 (ksi)	7.90	7.94*	0.5	1.34	1.33*	0.7

*Transformed stresses in to fiber and matrix directions to be consistent with FEMAP output vectors

**FEA results for this layup are based on a laminate with thin width (0.1 in.)

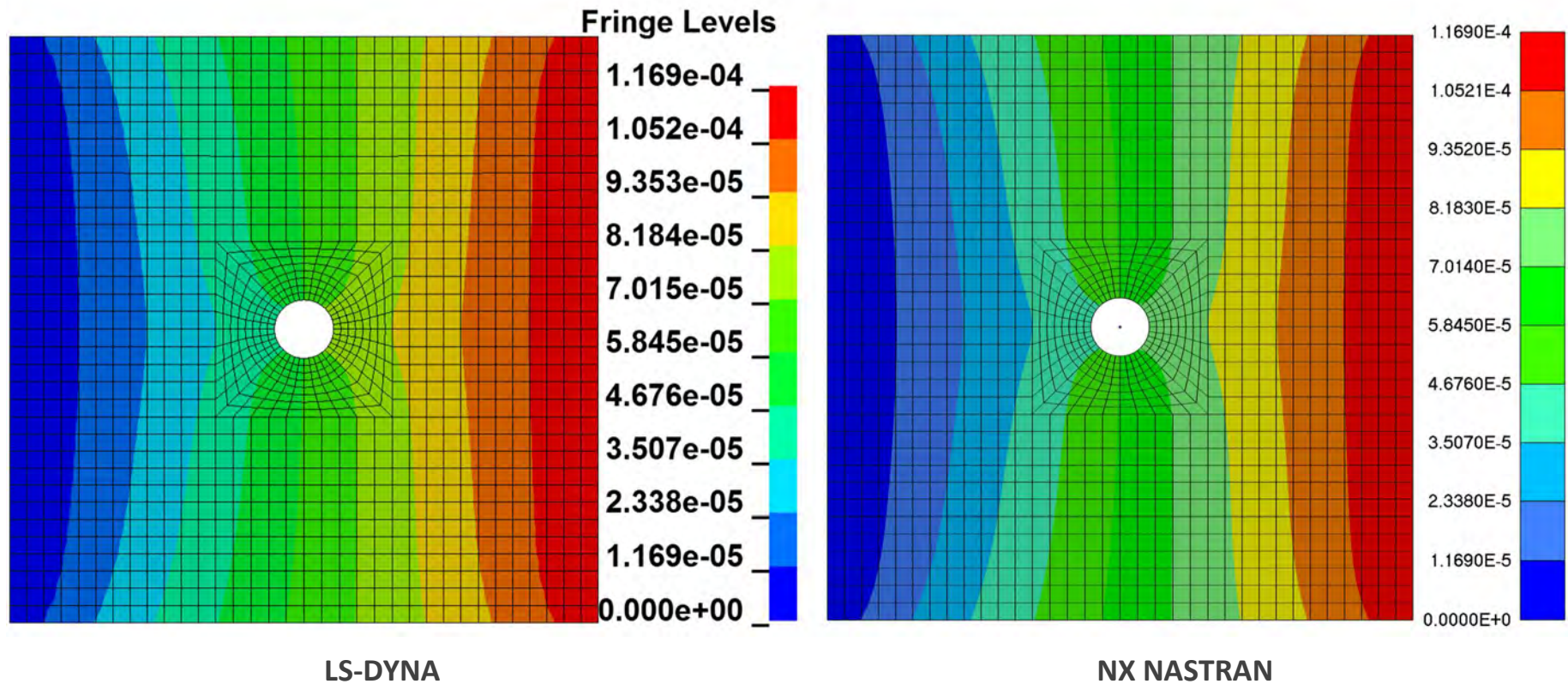
Note: ASTM Standard D7249 is often used for evaluating the flexural behavior of sandwich composites. This standard provides equations for calculating the stresses in the facesheet corresponding to a maximum load. However, it should be noted that the standard is not ideal for calculating stresses in the facesheet with different ply orientations. Rather, the standard is applicable for facesheets with unidirectional ply orientation and homogeneous facesheets (e.g. aluminum).

12. ADDENDUM

In this section, the examples presented in the white paper with the NX Nastran Solver are replicated using the LS-DYNA solver. FEMAP is the preprocessor however some modifications were done in LS-PrePost for TSHELL (LS-DYNA's layered solid) elements. The following is the summary of results using the LS-DYNA solver. A brief comparison of the results was also presented to understand the applicability of both the solvers.

12.1 EXAMPLE 1 (2D LAMINATE WITH A HOLE)

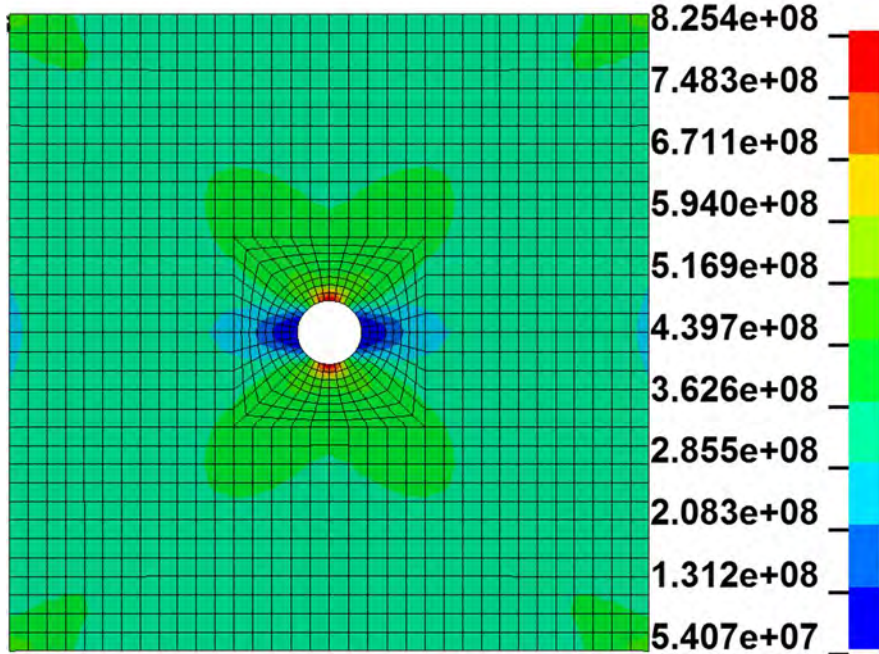
12.1.1 DISPLACEMENT CONTOUR



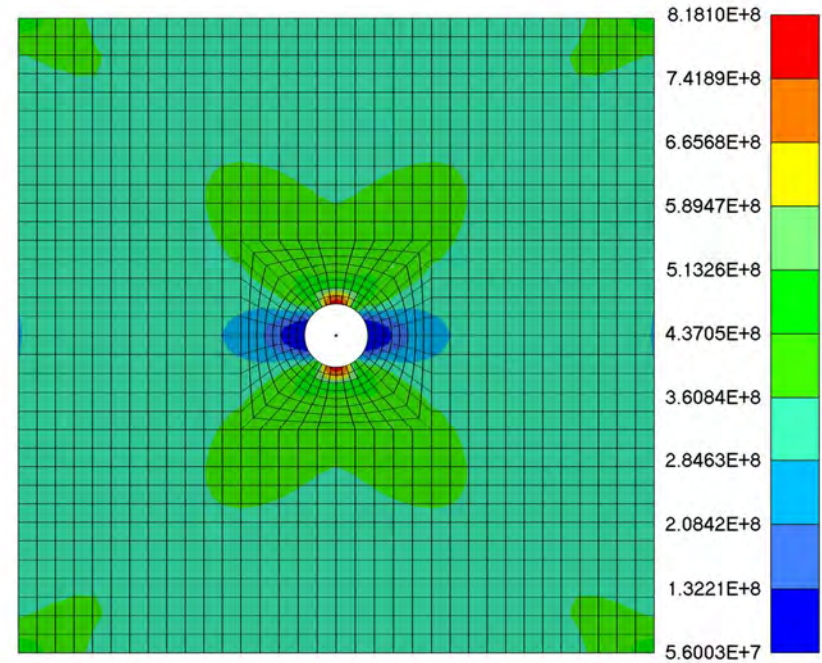
Remark: Displacement plots match quite well

12.1.2 MAJOR PRINCIPAL STRESS IN PLY 4 (0°)

Fringe Levels



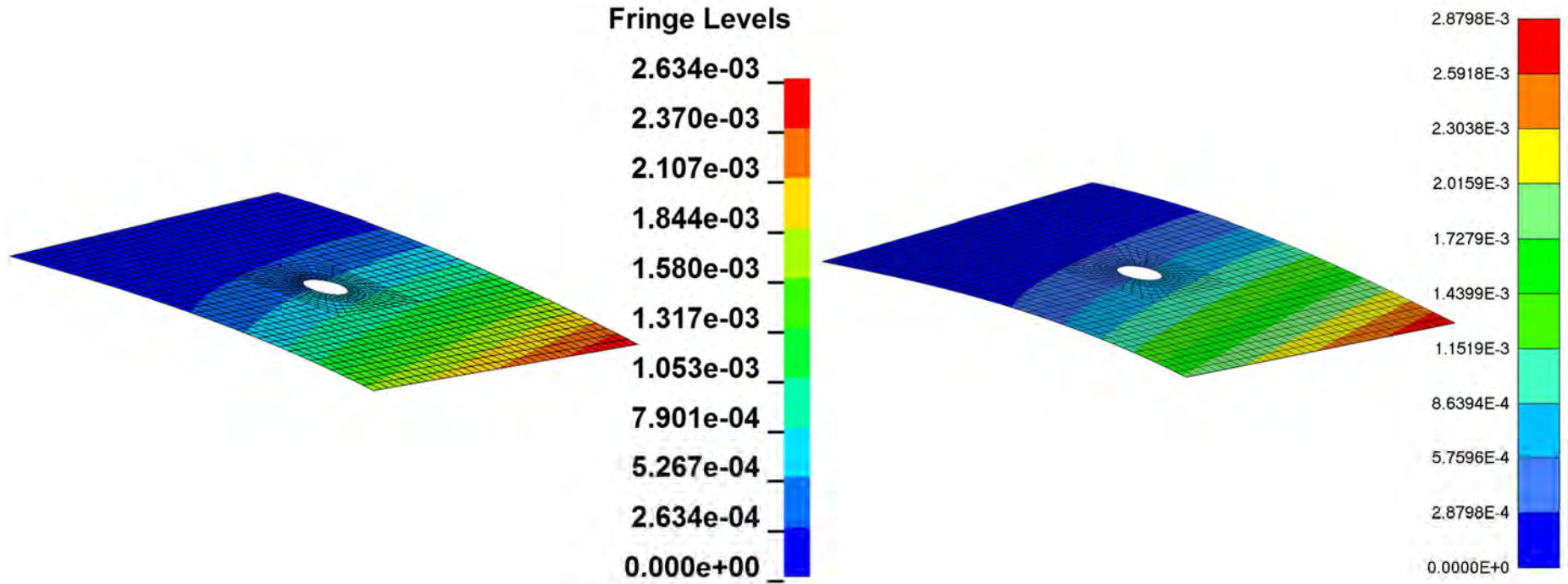
LS-DYNA



NX NASTRAN

Remark: Major Principal Stress plots match well. In order to analyze ply level stresses, the number of integration points (plies) in the laminate has to be entered in DATABASE_EXTENT_BINARY (MAXINT=8 for this example).

12.1.3 UNSYMMETRIC LAMINATE



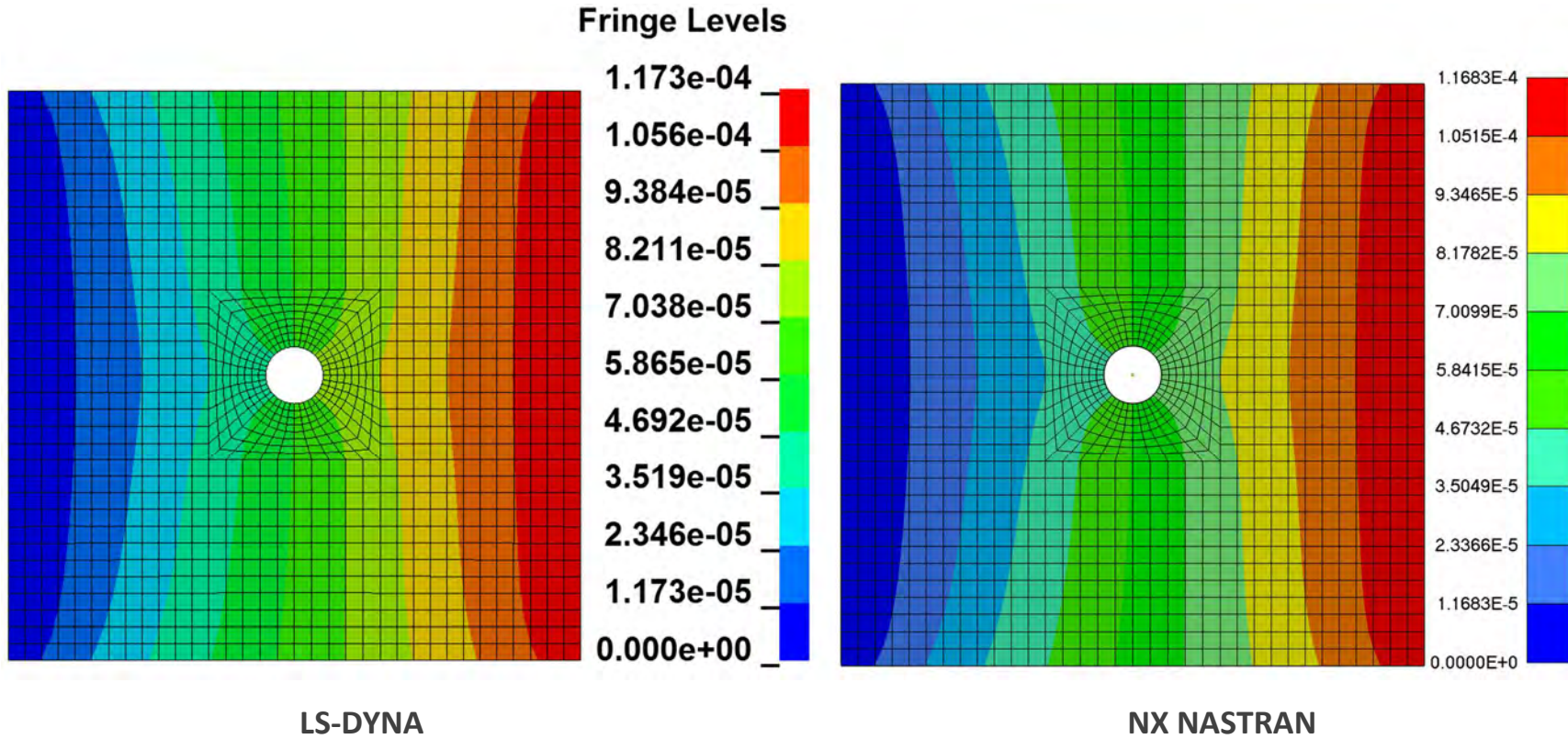
LS-DYNA

NX NASTRAN

Remark: The out-of-plane deformations observed from both the solvers are similar. Although this example uses shell elements, when modeling an unsymmetric laminate in LS-DYNA, the moduli and Poisson’s ratio in direction-3 (out-of-plane) should also be entered. Otherwise, erroneous results can be encountered. The reasoning behind the out-of-plane deformations observed in an unsymmetric laminate despite the uniaxial load is detailed in the Appendix.

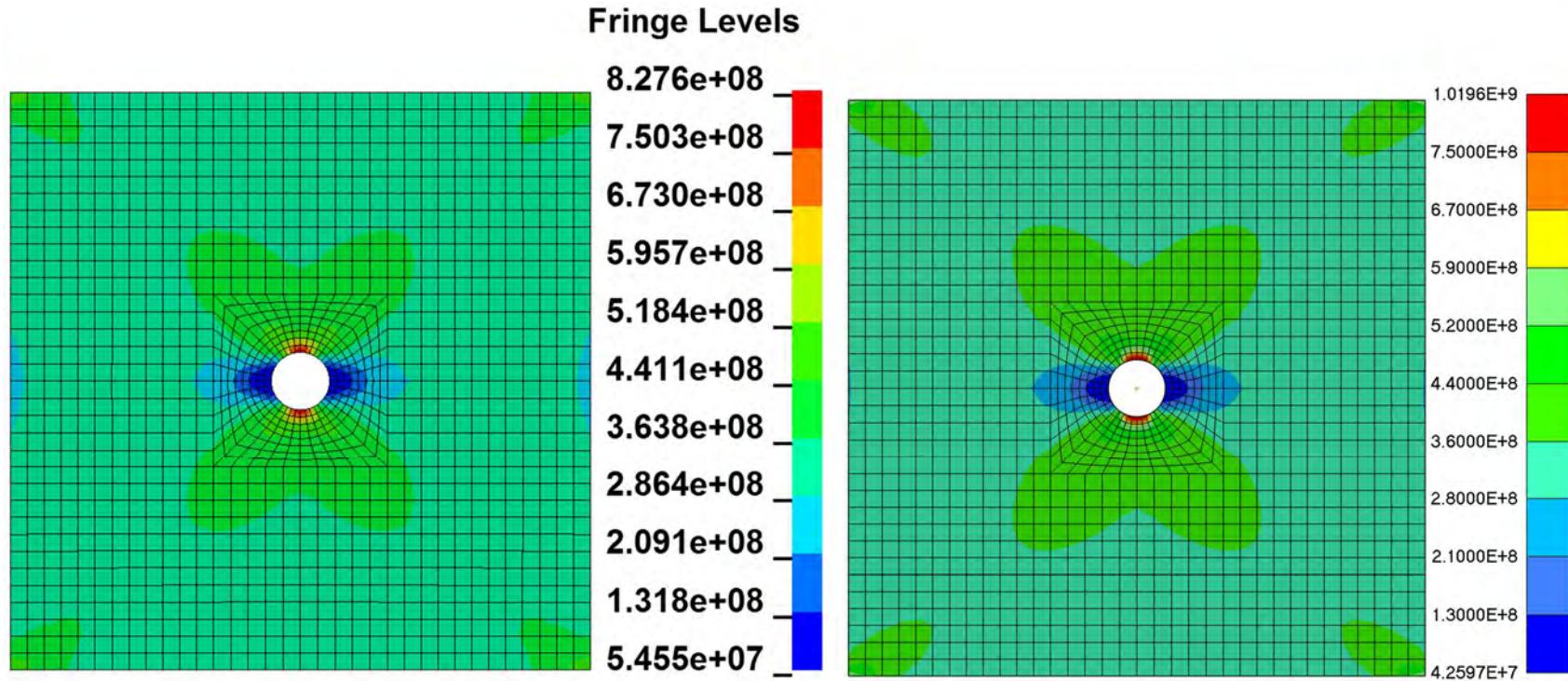
12.2 EXAMPLE 2 (3D LAMINATE WITH A HOLE)

12.2.1 DISPLACEMENT CONTOUR



Remark: Displacement plots match quite well

12.2.2 MAJOR PRINCIPAL STRESS IN PLY 4 (0°)

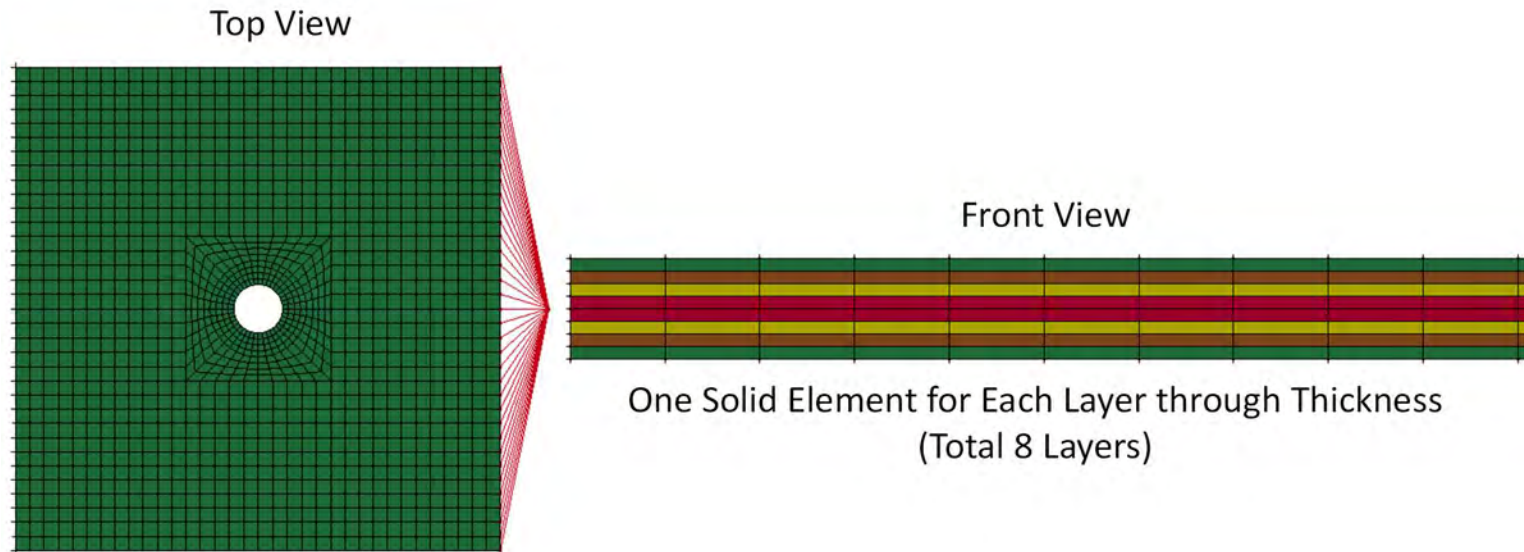


LS-DYNA

NX NASTRAN

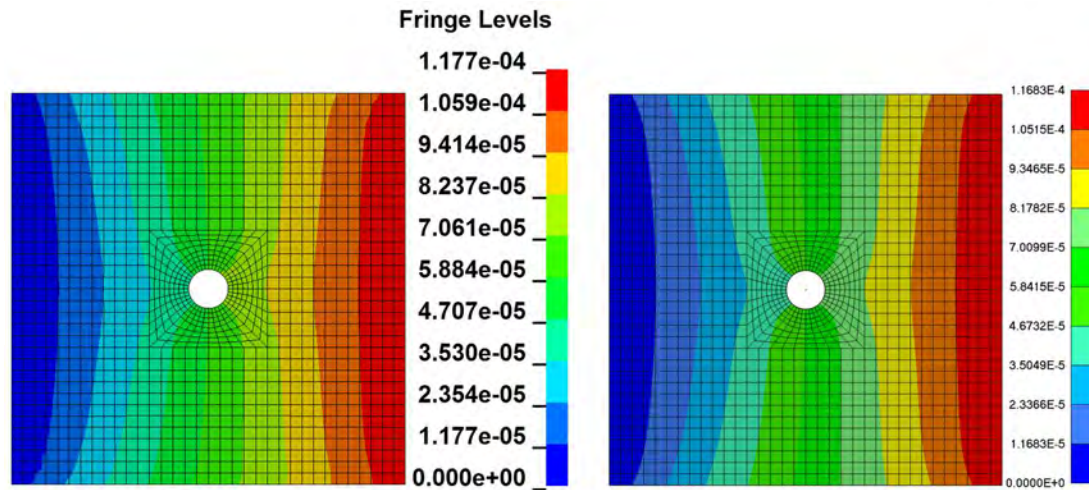
Remark: The contour of the major principal stress for ply 4 looks similar between LS-DYNA and NX Nastran. However, the results deviate (max result varies by ~19%). The LS-DYNA result with TSHELL (thick shell) was observed to be similar to shell formulation (example 1).

12.2.3 3D SOLID LAMINATE WITH ONE ELEMENT FOR EACH LAYER THROUGH THICKNESS

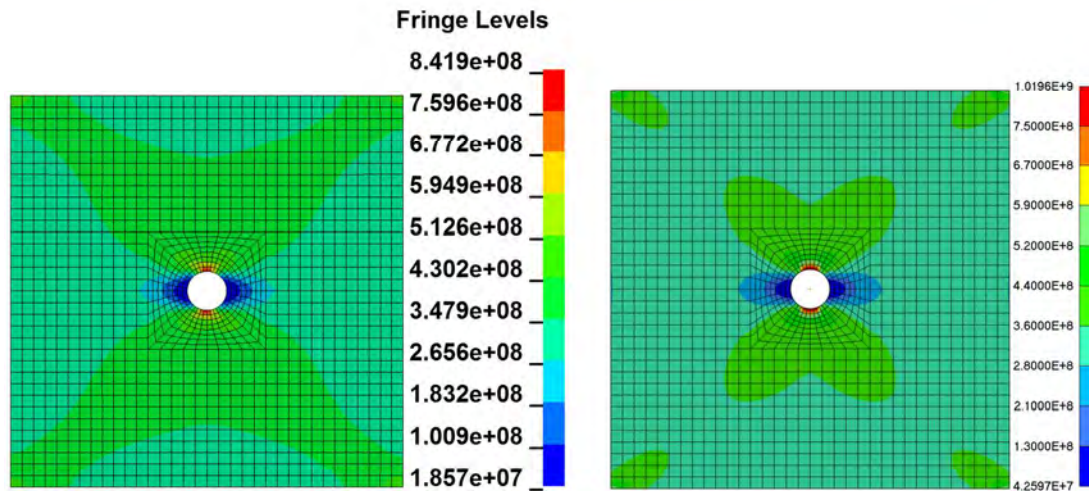


Remark: Here, one solid element for each ply is explicitly defined as shown above. This is a computationally expensive procedure which is particularly significant when modeling composite laminates with several layers (e.g., >50). This led to the development of a TSHELL element (discussed previously) using which all the plies could be defined in a single element layer through the laminate thickness.

Displacement contour



Major Principal Stress in Ply 4 (0°)



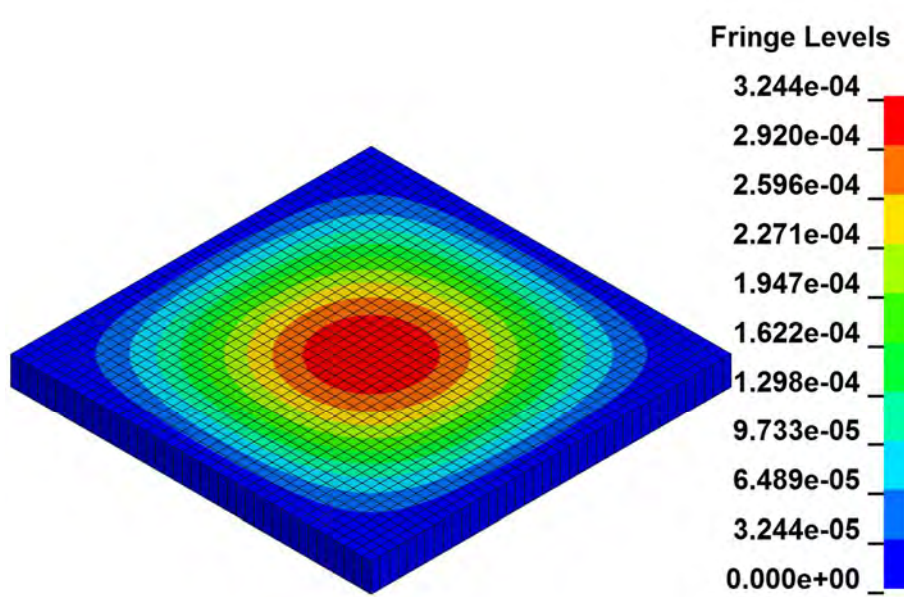
LS-DYNA

NX NASTRAN

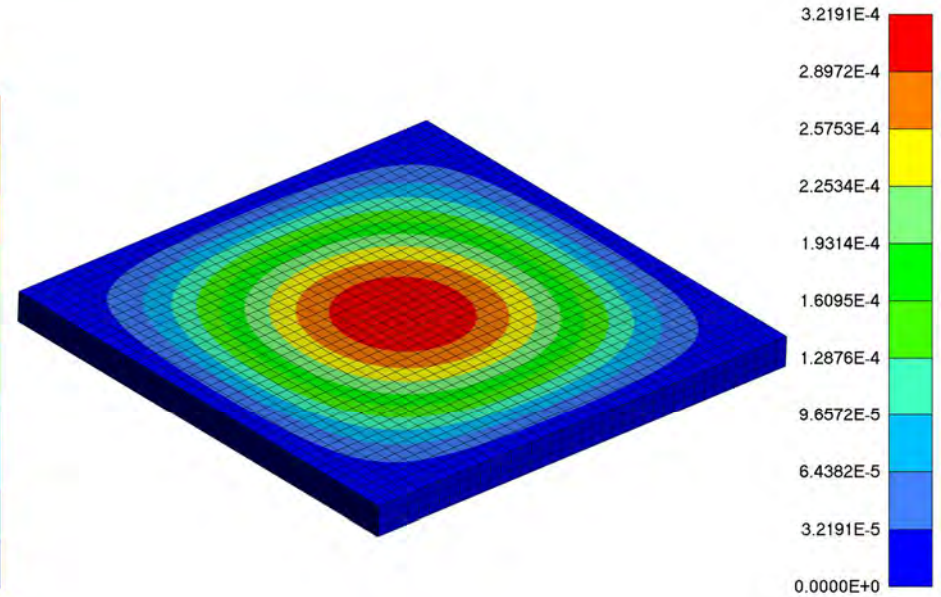
Remark: For this example, the displacement and stress plots are similar between the regular solid element and NX Nastran's Laminate Solid element.

12.3 EXAMPLE 3 (3D SANDWICH COMPOSITE)

12.3.1 DISPLACEMENT CONTOUR



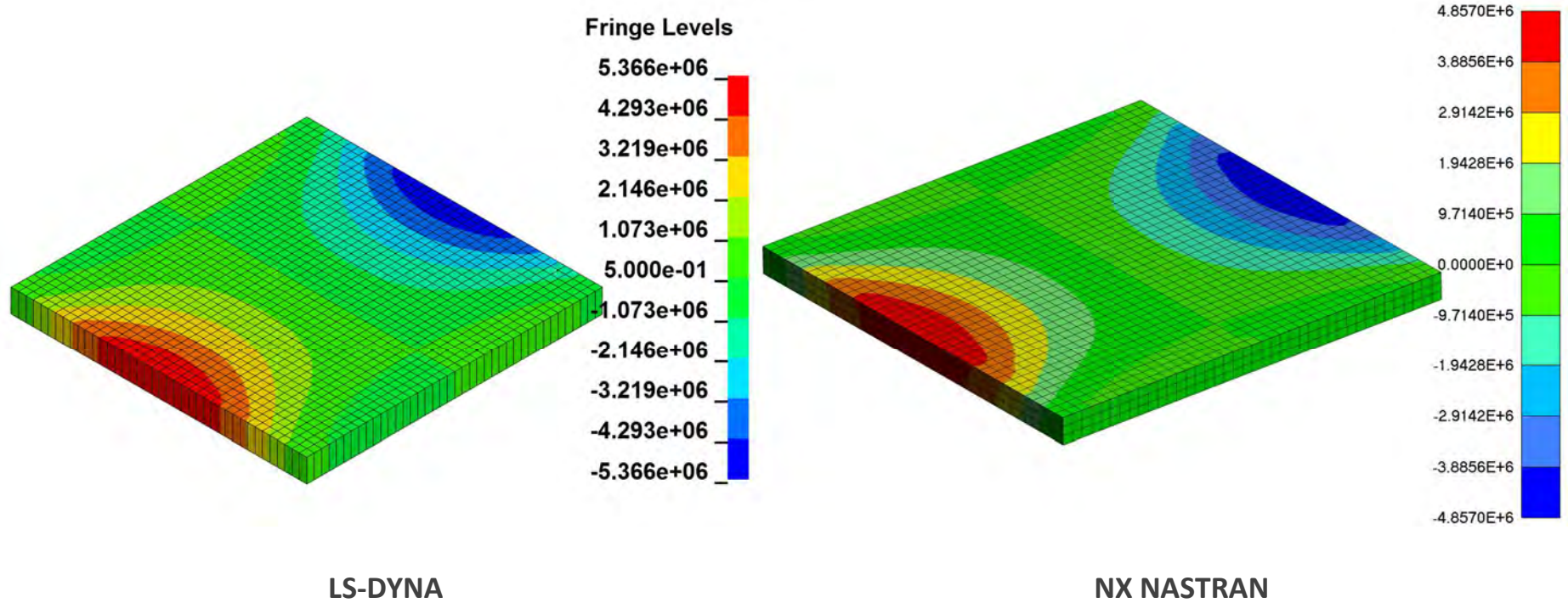
LS-DYNA



NX NASTRAN

Remark: Displacement plots match well

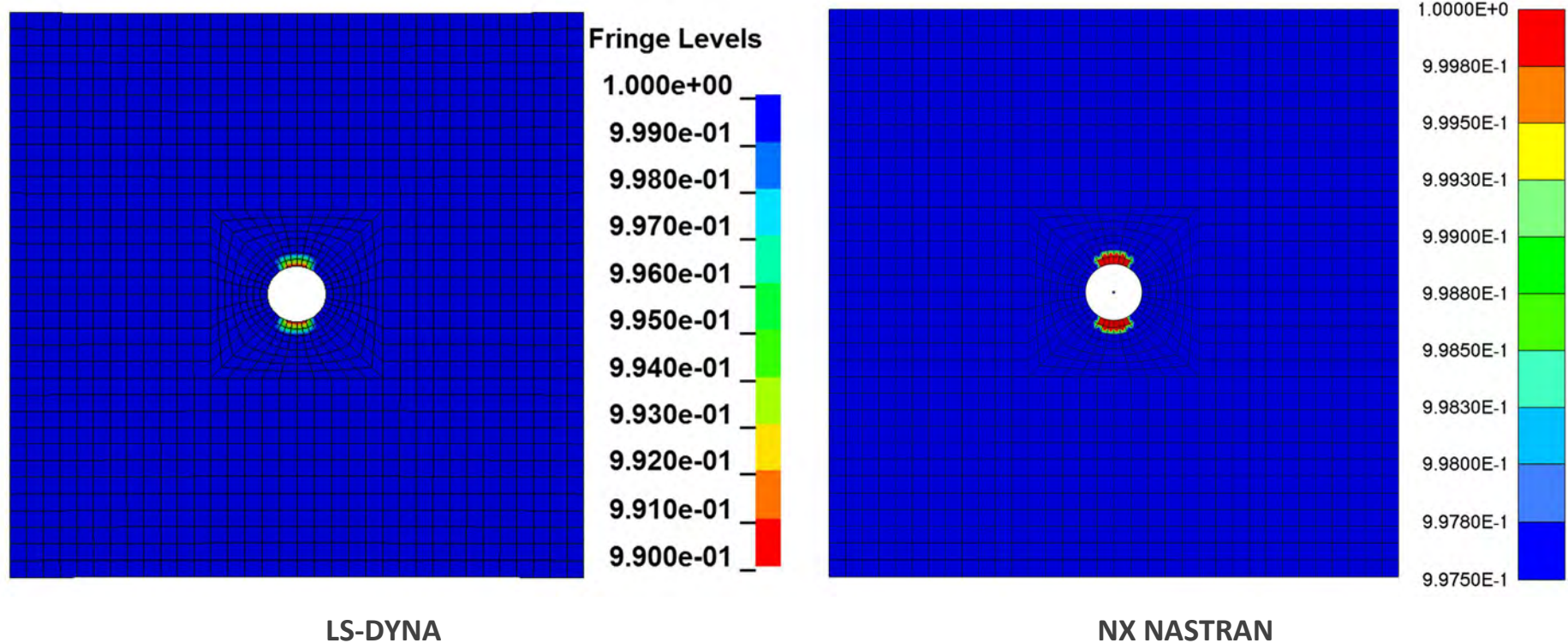
12.3.2 TRANSVERSE SHEAR STRESS (ZX) IN CORE



Remark: The transverse shear stress plots on the core look similar however they vary by ~10%.

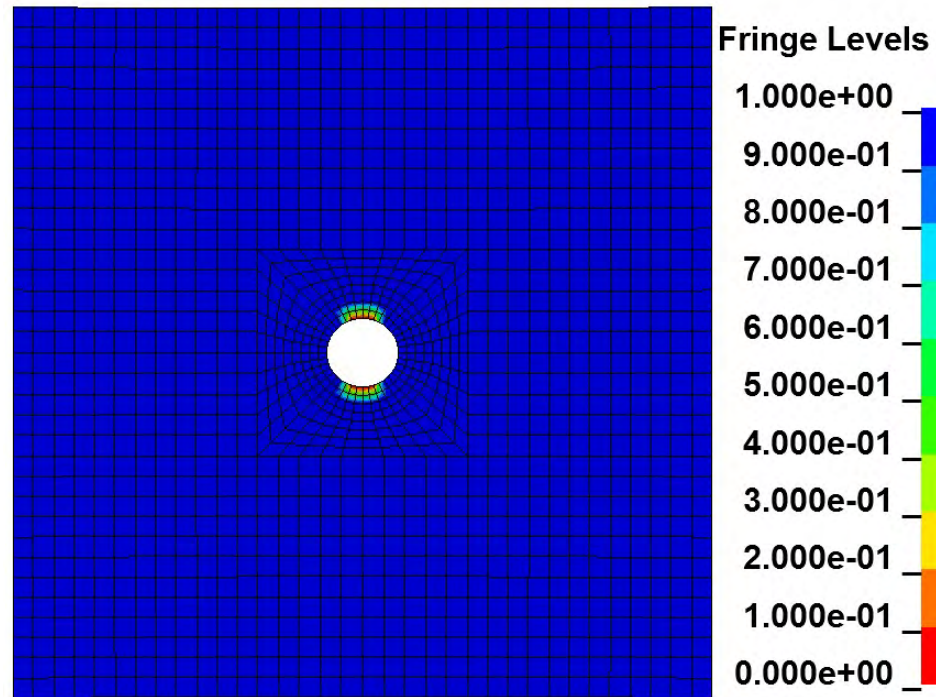
12.4 EXAMPLE 4 (2D LAMINATE WITH FAILURE)

12.4.1 LAMINATE FAILURE



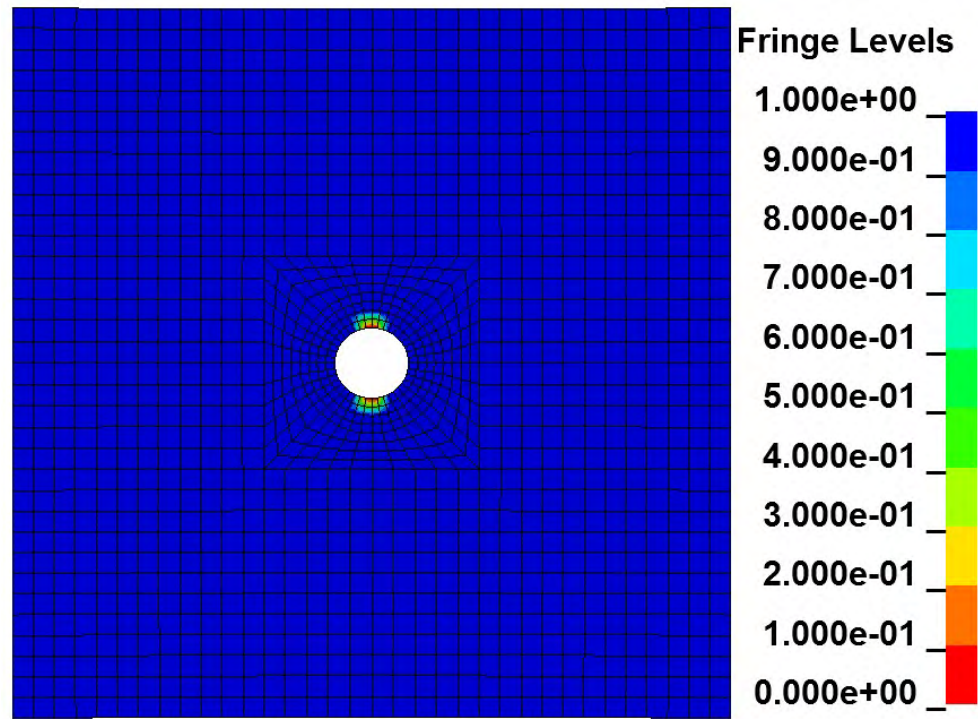
Remark: The failure region predicted by both the solvers is similar. In LS-DYNA a flag=1 implies no failure and flag=0 implies complete failure. We see that in the regions other than red, failure has initiated in some plies. The minimal value is not zero implying that the element did not fail completely. This means that in some plies, for the same element, failure did not occur. For ply level failures, LS-DYNA can explicitly specify the mode of failure whereas in NX Nastran we saw that it only indicates if a ply has failed or not (but not the failure mode). In order to analyze ply level failures (failure modes), additional history variables need to be requested in addition to specifying the total number of integration points. The NEIPS parameter in DATABASE_EXTENT_BINARY is set to 6 for this example. The output vector corresponding to each of these history variables is listed in the LS-DYNA manuals. For this example, we have used MAT_54 as our material model. History variable 5 (for MAT_54) gives the laminate failure index.

12.4.2 FAILURE IN LONGITUDINAL TENSION



Remark: Ply 4 and Ply 5 (0°) have partially failed in longitudinal tension. This is output using history variable 1.
 None of the plies have failed in longitudinal compression (history variable 2).

12.4.3 FAILURE IN TRANSVERSE TENSION



Remark: Plies 1, 8 (45°) and Plies 3, 6 (-45°) have partially failed in transverse tension. This is output using history variable 3.

None of the plies have failed in transverse compression (history variable 4).

13. APPENDIX

13.1 CLASSICAL LAMINATION THEORY

The classical lamination theory (CLT) is similar to the classical plate theory (CPT), the only difference is that CPT assumes that the material is isotropic, while the CLT is an extension to fiber-reinforced composite laminates with multiple layers resulting in complicated stress-strain relations [6]. Similar to the Euler-Bernoulli beam theory and the plate theory, the CLT is only valid for thin laminates (span and width > 10×thickness) with small displacements in the transverse direction.

The four cornerstones of the CLT are the kinematic, constitutive, force resultant, and equilibrium equations.

13.1.1 KINEMATIC EQUATIONS

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + z \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u_0}{\partial x} \\ \frac{\partial v_0}{\partial y} \\ \frac{\partial u_0}{\partial y} + \frac{\partial v_0}{\partial x} \end{Bmatrix} - z \begin{Bmatrix} \frac{\partial^2 w_0}{\partial x^2} \\ \frac{\partial^2 w_0}{\partial y^2} \\ 2 \frac{\partial^2 w_0}{\partial x \partial y} \end{Bmatrix}$$

Here, u_0 , v_0 , and w_0 are the displacements of the middle plane in the x , y , and z directions. Similarly, ε_x^0 , ε_y^0 , and γ_{xy}^0 are the mid-plane strains, and κ_x , κ_y , and κ_{xy} are the curvatures.

13.1.2 CONSTITUTIVE EQUATIONS

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}_k = [T]_k^T [S]_k [T]_k = \begin{bmatrix} \bar{S}_{11} & \bar{S}_{12} & \bar{S}_{16} \\ & \bar{S}_{22} & \bar{S}_{26} \\ sym. & & \bar{S}_{66} \end{bmatrix}_k \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix}_k$$

where the subscript k indicates the k^{th} layer counting from the top of the laminate.

13.1.3 RESULTANTS

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \int_{-t/2}^{t/2} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz = \sum_{k=1}^N \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}_k dz$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \int_{-t/2}^{t/2} z \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz = \sum_{k=1}^N \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}_k z dz$$

The terms on the left hand side are force and moment resultants.

13.1.4 EQUILIBRIUM EQUATIONS

$$\frac{\partial N_x}{\partial x} + \frac{\partial N_{yx}}{\partial y} = -p_x$$

$$\frac{\partial N_{xy}}{\partial x} + \frac{\partial N_y}{\partial y} = -p_y$$

$$\frac{\partial^2 M_x}{\partial x^2} + 2 \frac{\partial^2 M_{xy}}{\partial x \partial y} + \frac{\partial^2 M_y}{\partial y^2} = -p_z$$

13.1.5 STIFFNESS MATRICES A, B, AND D

The relations between the resultants (forces N and moments M) and the strains (strains ϵ and curvatures κ) are of interest in composite laminates. By replacing the stresses in the force and moment resultants with strains via the constitutive equations, we have

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \sum_{k=1}^N \begin{bmatrix} \bar{C}_{11} & \bar{C}_{12} & \bar{C}_{16} \\ & \bar{C}_{22} & \bar{C}_{26} \\ \text{sym.} & & \bar{C}_{66} \end{bmatrix}_k \left(\int_{z_{k-1}}^{z_k} dz \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + \int_{z_{k-1}}^{z_k} z dz \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix} \right)$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \sum_{k=1}^N \begin{bmatrix} \bar{C}_{11} & \bar{C}_{12} & \bar{C}_{16} \\ & \bar{C}_{22} & \bar{C}_{26} \\ \text{sym.} & & \bar{C}_{66} \end{bmatrix}_k \left(\int_{z_{k-1}}^{z_k} z dz \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + \int_{z_{k-1}}^{z_k} z^2 dz \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix} \right)$$

By applying the summation and integration operations to their respective components, the force and moment resultants can be expressed as,

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ & A_{22} & A_{26} \\ \text{sym.} & & A_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ & B_{22} & B_{26} \\ \text{sym.} & & B_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ & B_{22} & B_{26} \\ \text{sym.} & & B_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ & D_{22} & D_{26} \\ \text{sym.} & & D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix}$$

The above equations can be expressed as,

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

where A is called the extensional stiffness, B is called the coupling stiffness, and D is called the bending stiffness of the laminate. The components of these three stiffness matrices are defined as follows:

$$A_{ij} = \sum_{k=1}^N (\bar{C}_{ij})_k (z_k - z_{k-1}) = \sum_{k=1}^N (\bar{C}_{ij})_k t_k$$

$$B_{ij} = \frac{1}{2} \sum_{k=1}^N (\bar{C}_{ij})_k (z_k^2 - z_{k-1}^2) = \sum_{k=1}^N (\bar{C}_{ij})_k t_k \bar{z}_k$$

$$D_{ij} = \frac{1}{3} \sum_{k=1}^N (\bar{C}_{ij})_k (z_k^3 - z_{k-1}^3) = \sum_{k=1}^N (\bar{C}_{ij})_k \left(t_k \bar{z}_k^2 + \frac{t_k^3}{12} \right)$$

Here t_k is the thickness of the k^{th} layer and \bar{z}_k is the distance from the mid-plane to the centroid of the k^{th} layer.

13.2 UNSYMMETRIC LAYUP

From the classical lamination theory, we have the following relation between force/moment resultants, and the strains and curvatures.

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ & A_{22} & A_{26} \\ sym. & & A_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ & B_{22} & B_{26} \\ sym. & & B_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ & B_{22} & B_{26} \\ sym. & & B_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} + \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ & D_{22} & D_{26} \\ sym. & & D_{66} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix}$$

For example consider only the axial force component N_x . We have,

$$N_x = A_{11}\varepsilon_x^0 + A_{12}\varepsilon_y^0 + A_{16}\gamma_{xy}^0 + B_{11}\kappa_x + B_{12}\kappa_y + 2B_{16}\kappa_{xy}$$

For an isotropic material, we have only A_{11} and A_{12} . The remaining components A_{16} , B_{11} , B_{12} , and B_{16} are all zero. Thus, the axial force component in an isotropic material is related to inplane strains only. However, for composite laminates,

the components A_{16} , B_{11} , B_{12} , and B_{16} can exist and will influence the deformation behavior. The ‘compute’ option in layup editor can be handy in checking these components.

For the symmetric laminate in Example 1, the following information is obtained when the ‘compute’ option is selected. We can see that all the components in the B matrix (coupling stiffness matrix) are zero. Thus, we do not see any out-of-plane deformation in the symmetric laminate case.

	A Matrix		
9.48215E+7	3.01204E+7	0.00000E+0	
3.01204E+7	9.48215E+7	0.00000E+0	
0.00000E+0	0.00000E+0	3.23506E+7	
	B Matrix		
0.00000E+0	0.00000E+0	0.00000E+0	
0.00000E+0	0.00000E+0	0.00000E+0	
0.00000E+0	0.00000E+0	0.00000E+0	
	D Matrix		
1.09875E+1	8.07952E+0	5.17406E+0	
8.07952E+0	2.34052E+1	5.17406E+0	
5.17406E+0	5.17406E+0	8.53069E+0	

Similarly, for the unsymmetric laminate,

	A Matrix		
9.48215E+7	3.01204E+7	0.00000E+0	
3.01204E+7	9.48215E+7	0.00000E+0	
0.00000E+0	0.00000E+0	3.23506E+7	
	B Matrix		
-1.3176E+4	2.54946E+3	5.31327E+3	
2.54946E+3	8.07707E+3	5.31327E+3	
5.31327E+3	5.31327E+3	2.54946E+3	
	D Matrix		
2.12521E+1	6.09339E+0	1.03481E+0	
6.09339E+0	1.71129E+1	1.03481E+0	
1.03481E+0	1.03481E+0	6.54455E+0	

Now, we see that the B matrix has finite values and as a result, we see out-of-plane deformation in the unsymmetric laminate case. This information is quite useful in designing unsymmetric laminates, by tailoring the layup orientation angles to retain required matrix components (ABD matrices) only.

13.3 CHAMIS MODEL

$$E_{11} = V_f \cdot E_{11_f} + V_m \cdot E_m$$

$$E_{22} = \frac{E_m}{1 - \sqrt{V_f} \cdot \left(1 - \frac{E_m}{E_{22_f}}\right)}$$

$$\nu_{12} = V_f \cdot \nu_{12_f} + V_m \cdot \nu_m$$

$$G_{12} = \frac{G_m}{1 - \sqrt{V_f} \cdot \left(1 - \frac{G_m}{G_{12_f}}\right)}$$

13.4 ONSET FAILURE THEORY

The onset theory [7-9] assumes that the critical effective properties of the lamina that control damage initiation are the effective volumetric and equivalent strains of the lamina. Both the volumetric strain and the equivalent strain of the lamina are defined using strain invariants. The volumetric strain (critical dilatational strain invariant) is defined by the first invariant of the strain and the equivalent strain (critical distortional strain invariant) is a function of the second invariant of the strain. The strain invariants are functions of three principal strains. The volumetric strain is given by,

$$\text{Volumetric strain} = J_1 + J_2 + J_3 \tag{1}$$

Where,

$$J_1 = \varepsilon_1 + \varepsilon_2 + \varepsilon_3$$

$$J_2 = \varepsilon_1 \varepsilon_2 + \varepsilon_1 \varepsilon_3 + \varepsilon_2 \varepsilon_3$$

$$J_3 = \varepsilon_1 \varepsilon_2 \varepsilon_3$$

ε_1 , ε_2 , and ε_3 are the three principal strains

J_1 , J_2 , and J_3 are the invariants of the strain tensor

The significant component of the volumetric strain is the first invariant of strain tensor. The reduced form of volumetric strain is given by,

$$\text{Volumetric strain} = J_1 = \varepsilon_1 + \varepsilon_2 + \varepsilon_3 \quad (2)$$

The equivalent strain (distortional strain invariant, change in shape) is given by,

$$\varepsilon_{eqv} = \sqrt{\frac{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_1 - \varepsilon_3)^2}{2}} \quad (3)$$

For details on the implementation of the onset theory, it is recommended to read the references [7-9].

REFERENCES

1. H. Altenbach, J. Altenbach, and W. Kissing, "Mechanics of Composite Structural Elements," Springer-Verlag Berlin Heidelberg, New York, 2004.
2. <http://structuralmechanics.com/about/resources/newsletter/articles/you-pull-it-twists-tailored-composites/>
3. R. Seemann and D. Krause, "Numerical Modeling of Nomex Honeycomb Cores for Detailed Analyses of Sandwich Panel Joints," 11th World Congress on Computational Mechanics, pp. 1-12, Barcelona, Spain, July 20-25, 2014.
4. http://www.engineeringtoolbox.com/polymer-composite-fibers-d_1226.html
5. <http://ocw.mit.edu/courses/materials-science-and-engineering/3-11-mechanics-of-materials-fall-1999/modules/props.pdf>
6. http://www.efunda.com/formulae/solid_mechanics/composites/comp_laminate.cfm
7. J. Gosse and S. Christensen, "Strain Invariant Failure Criteria for Polymers in Composite Materials," 42nd AIAA Structures, Structural Dynamics, and Materials Conference and Exhibit, Seattle, WA, pp. 1-11, 2001.

8. L. Hart-Smith, "Mechanistic Failure Criteria for Carbon and Glass Fibers Embedded in Polymers in Polymer Matrices," 42nd AIAA Structures, Structural Dynamics, and Materials Conference and Exhibit, Seattle, WA, pp. 1-11, 2001.
9. S. McNaught, "Implementation of the Strain Invariant Failure Theory for Failure of Composite Materials," Thesis, October 2009.
10. S. Tsai and H. Hahn, "Introduction to Composite Materials," Technomic Publishing Company, Westport, Connecticut, 1980.

Equivalent laminate models are useful when only displacements, buckling loads and modes, or vibration frequencies and modes are required (Barbero book).