Study of Forming of Composite Materials with Abaqus CAE and The Preferred Fiber Orientation (PFO) Model

Thesis

Presented in Partial Fulfillment of the Requirements for the Degree Master of Science in the Graduate School of The Ohio State University

By

Yumeng Li, B.S.

Graduate Program in Mechanical Engineering
The Ohio State University
2017

Thesis Committee

Farhang Pourboghrat
Prasad Mokashi
Copyrighted by

Yumeng Li

2017
Abstract

The forming of composite materials may lead to fiber angle change during the deformation. The change of fiber orientations can lead to changes in mechanical properties of the composite material. Therefore, it is important to know the changes of the fiber orientations in order to calculate effective material properties of the material. The constitutive model for obtaining properties of the composite material has been investigated in this study. Zampaloni originally developed the preferred fiber orientation (PFO) model in his PhD study, in which he tracked the fiber orientation during the composite forming process. He showed that PFO model gives more accurate results compared with the Abaqus/CAE model. The simple tension and shear tests were studied with both Abaqus simulation software, and analytical calculations. Results from both models were compared, and it was shown that compared with Abaqus, the PFO model tracks the fiber angle correctly. The stamping of a hat shape section was also simulated with the PFO model and simulation results were compared with experiments. A good agreement between simulation results and experimental results were obtained, which suggests that the PFO model can predict fiber angle change correctly, and its results are closer to the real forming situation.
Dedication

I dedicate this work to my parents. Thanks for the love, support and guidance.
Acknowledgments

Thank you to everyone who has helped me with research in my graduate study.

Thanks to Dr. Hyunchul Ahn for his expertise in composite materials.
Most of all, thanks to my advisor, Prof. Farhang Pourboghrat, for this opportunity and support throughout my graduate study.
Vita

December 2015………………. B.S. Mechanical Engineering, The Ohio State University
2015 to present………………..M.S. Mechanical Engineering, The Ohio State University

Fields of Study

Major Field: Mechanical Engineering
Table of Contents

Abstract ........................................................................................................................................ ii
Dedication ...................................................................................................................................... iii
Acknowledgments ..................................................................................................................... iv
Vita .................................................................................................................................................. v
List of Tables .................................................................................................................................. vii
List of Figures ............................................................................................................................... viii
Chapter 1. Introduction ............................................................................................................... 1
  Forming of Composite Materials ............................................................................................... 6
Chapter 2. Literature Review ...................................................................................................... 9
  Modeling Techniques .................................................................................................................. 9
  Material Characterization Methods ........................................................................................... 12
Chapter 3. Numerical Methods .................................................................................................. 16
  Constitutive Model .................................................................................................................... 16
  Obtaining Material Properties .................................................................................................. 19
  Preferred Orientations ............................................................................................................... 21
  Implementation into Abaqus ....................................................................................................... 25
Analytical & Simulation Discussion ............................................................................................. 27
Numerical Validation .................................................................................................................. 32
  Numerical Setup ...................................................................................................................... 32
PFO Validation ............................................................................................................................ 41
  Validation with experiments ....................................................................................................... 45
Conclusions .................................................................................................................................. 55
References ...................................................................................................................................... 57
List of Tables

Table 1. Properties of composite ................................................................. 27
List of Figures

Figure 1. Two types of reinforcement for fibrous composite [3] ........................................ 3
Figure 2. Several types of particulate composite [3] .......................................................... 4
Figure 3. Composition of a laminated composite [4] ........................................................... 5
Figure 4. Match die molding [7] ....................................................................................... 7
Figure 5. Material frame and global coordinate [7] ............................................................. 17
Figure 6. Stiffness summation in PFO model [7] ................................................................. 22
Figure 7. Rotation of PFO to the global coordinate [7] ....................................................... 23
Figure 8. Effects of deformation tensor on elements [7] ..................................................... 25
Figure 9. Flow chart of PFO model implemented in Abaqus/CAE [9] ............................... 26
Figure 10. Modulus plot for single layer ............................................................................ 28
Figure 11. Properties of a single layer with 0 degree ....................................................... 29
Figure 12. Young’s modulus plot in fiber direction ......................................................... 30
Figure 13. Young’s modulus in transverse direction ....................................................... 30
Figure 14. Shear modulus ............................................................................................... 31
Figure 15. Dimension of a composite specimen ............................................................... 32
Figure 16. Material properties of the specimen ............................................................... 33
Figure 17. Shell composite section of the specimen ......................................................... 34
Figure 18. Material direction of the specimen ............................................................... 34
Figure 19. Bottom boundary condition of the specimen ................................................ 35
Figure 20. Top boundary condition of the specimen ....................................................... 35
Figure 21. Result of the tensile test .................................................................................. 36
Figure 22. Stress vs. strain curve for [45/-45/45/-45/45/-45/45]s ..................................... 37
Figure 23. Stress vs. strain curve for [90/0/90/0/90] in its fiber direction ...................... 37
Figure 24. Stress vs. strain curve for [90/0/90/0/90] in its transverse direction .......... 38
Figure 25. Simple shear test ............................................................................................. 39
Figure 26. Calculation of shear modulus ......................................................................... 39
Figure 27. Fiber orientation after deformation of the tension test .................................. 41
Figure 28. Tension test result of fiber angle (degrees) after deformation with PFO model ................................................................................................................. 42
Figure 29. Coordinates of deformed element with PFO model ........................................ 43
Figure 30. Result of shear test with PFO model ............................................................... 44
Figure 31. Calculation of fiber orientation for shear test ................................................ 44
Figure 32. Undeformed shape of uniaxial extension test ................................................ 46
Figure 33. Deformed shape of uniaxial extension test ..................................................... 46
Figure 34. Deformed preform at elongation of 25 % ....................................................... 47
Figure 35. Three different modes of deformation in experiment ........................................ 48
Figure 36. Geometry for upper die .................................................................................. 49
Figure 37. Assembly of the stamping simulation ............................................................ 50
Figure 38. Stress distribution of the sheet (0/90 degrees) ............................................... 50
Figure 39. Result of stamping process of the sheet (0/90 degrees), SDV5 is the updated fiber angle after deformation in degrees ................................................................. 51
Figure 40. Stamped hat section from experiment (0/90 degrees) ..................................... 52
Figure 41. Stamping simulation with 45/-45 PFO. SDV5 represents fiber angle orientations after the deformation in degree ................................................................. 53
Chapter 1. Introduction

The interest in using composite materials has been on the rise for the last couple of decades, since composites offer considerable high strength to weight ratio. The modern use of composite materials, fiber-reinforced polymer composites especially, could be traced back to the World War II era, with most applications being rocket motor cases, and high-performance car bodies such as those for 1950’s Chevrolet Corvettes. Recently, in the aerospace industry, carbon fiber composite is widely used for its high strength, which could exceed that of steel, as well as its lower specific density. At first, the costs of the composite material were higher than traditional metal materials. Because of the higher costs of such new materials, engineering designers often had to make tough choices, since by using composite materials, the properties of such materials had to overcome the additional costs compared with conventional materials. As the industry grew, the cost of composite material decreased, and it is now commonly seen in consumer goods such as automotive components, sporting equipment, and even some high-end gaming laptops. Products like these are the first to utilize composite materials as it is becoming more affordable. However, the price of those products is still out of reach for average consumers. Nevertheless, the advantages of using composites are too attractive to ignore, especially for automotive and aerospace industries, where a material with strength as strong as steel and weight only a fraction of that of steel is needed. Many automotive components made
from composite materials can be seen in the market, as the automotive companies are striving for higher fuel efficiency. Since the 1960s, the composite market has been growing steadily at the rate of approximately 15% per year [1].

Composites can be categorized based on the type of reinforcement used in the material. Fibrous and particulate are two common classes. Each type of reinforcement has its own unique properties and application purposes, and can be subdivided into specific categories as following:

**Fibrous.** A fibrous composite consists of either continuous or chopped (whiskers) fibers suspended in a matrix material. They can be identified from a geometric viewpoint:

*Continuous Fibers.* A continuous fiber is characterized as having a very high length-to-diameter ratio based on its geometry. Usually they are stronger and stiffer than bulk material. Fiber diameters generally range between 0.00012 and 0.0074 μin (3-200 μm), depending upon the fiber [2].

*Whiskers.* A whisker is generally considered to be a short, stubby fiber. It is defined to have a length-to-diameter ratio of 5< l/d <1000 and beyond. Whisker generally have a diameter range between 0.787 and 3937 μin (0.02-100 μm)

Composites in which the reinforcements are discontinuous or whiskers can be produced to have reinforcements that have either random or biased orientations. Material systems having discontinuous reinforcements are considered single layer composites. Usually the discontinuities will produce anisotropic material response property, but the random orientation reinforcements have nearly isotropic material property in most situations. On the other hand, continuous fiber composites can have either single layer or
multiple layers. A single layer composite with continuous fiber reinforcement can be either unidirectional or woven, and multilayered composites with continuous fiber are commonly referred to as laminates. Generally, a continuous fiber composite is assumed to have orthotropic material property. A picture of both types of fibrous composites is shown in Figure 1.

![Figure 1. Two types of reinforcement for fibrous composite [3]](image)

**Particulate.** A particulate composite has particles suspended in a matrix. Particles can have random shape, size or configuration. The most common particulate composites in modern days are concrete and particle board. Two subclasses are included in this category: flake and filled/skeletal:

*Flake.* A flake composite usually has large ratios of platform area to thickness within the flakes, which is suspended in a resin system. (Similar to a particle board).

*Filled/Skeletal.* A filled/skeletal composite is composed of a continuous skeletal matrix filled by a second material: such as a honeycomb core filled with an insulating material.
The material response property of such composite can be either anisotropic or orthotropic. They can be used in the applications that do not require too much strength for design purposes. A picture of several types of particulate composite is shown in Figure 2 [3].

Figure 2. Several types of particulate composite [3]

In some applications, the fibers are aligned in certain directions to meet different strength requirements. The most common way is to use a material called “prepreg”. “Prepreg” is a general term for a reinforcing fabric composite, which has been pre-impregnated with a matrix system. The resin system already has proper curing agent, which means the “prepreg” could be directly laid into the mold with no additional resin. The “prepreg” resin is active, so it is better to keep the material in a freezer to suppress cure until it is needed for producing parts. When it is needed, the “prepreg” is taken out of the freezer and stored in room temperature, then cut into needed sizes, and laid up to a laminate, in which each lamina is oriented in the appropriate direction.
For the uncured laminate to be cured, elevated temperature and pressure are applied. Raising temperature could cause a chemical reaction to progress at a certain speed, and entrapped air within the laminate could be driven out by high pressure. The curing process is commonly done by an autoclave.

There are a lot of composites employing matrix systems such as polyester, vinylester and epoxy. And those kinds of plastics are known as thermoset, which are applied to the reinforcing fibers in liquid form. And heat or a chemical hardener is used to catalyze cross-linking of the polymer chains to harden the resin permanently. Low curing temperature, a low pre-cured viscosity, high stiffness and stability over a wide range of temperature are several advantages of such resin. However, it also has numerous disadvantages. The curing cycle of the resin system can take hours, which hinders the ability for mass production. Moreover, since thermosets are substantially infusible and insoluble, damaged components are not going to be fixed easily, and often they need to be replaced completely. Also, recycling resins is also difficult.

Some of the disadvantages can be overcome by using thermoplastic resin systems. This type of resin system softens and melts at high temperature and solidifies when cooled.
This will reduce the manufacturing cycle to a fraction of time when compared with those from thermosets. Also, thermoplastics are capable of being repeatedly softened and reshaped with temperature increase and hardened by temperature decrease. This is convenient for the repair and joining of components by fusion bonding and thermoplastic welding techniques [5]. In contrast to thermoset resins, thermoplastics do not have cross-linking; instead they form a solid structure by an entangled network of amorphous or semi-crystalline polymer chains. Such a structure gives thermoplastic polymer the ability to reach large deformations before failure, which leads to larger energy absorption than that from thermoset polymers. Also, thermoplastics are easily recycled, which benefits waste reduction [6].

However, thermoplastics also have several drawbacks that are worth mentioning. Often, thermoplastics need a higher processing temperature than thermosets. Take PEEK for an example, it requires approximately 340°C to melt and about 385°C to process. Moreover, poor interface adhesion between fiber and resin for thermoplastics lead to reduced mechanical properties. By adding a binding agent between the interface of fiber and matrix for consolidation can overcome those problems. Still, the benefits outweigh the cost for using thermoplastic matrix systems [7].

**Forming of Composite Materials**

The ability to be heated and reformed several times before the final shape has been a key feature for the thermoplastic composite. There are a lot of methods for forming composite materials with such matrix system. Match die molding is one of the most common methods of processing thermoplastic composites. It is similar to sheet metal
forming; instead it is composite material that is acted on by a die of the desired geometry. The composite is inserted between two dies, and the upper die closes on the composite material and pushes it into the bottom die, which is a mirror image of the upper die but slightly larger. When the two dies come together, the composite is pushed into the cavity that is used to determine the shape and thickness of the composite sheet. The process of match die molding is shown Figure 4.

Figure 4. Match die molding [7]

In this method, the dies need to be heated to the forming temperature for the composite material to deform, and then cooled to harden the material to its final shape. In this method, the heating and cooling of the tools are necessary for forming the required shape, however, it takes relatively long time for the die to be heated to the forming temperature and cooled for material to solidify. Therefore, this method is relatively time-
consuming. Also, the dies only exert downward force onto each other. For deep drawn part application, the normal directions may be significantly different from the force application vector along the die surface. Those differences will make it difficult to control the thickness of the part and maintain even pressure along the surface. And the final part might not have uniform thickness or be well consolidated.

Thermo hydro-forming can be utilized to form composite materials. This method uses heated pressurized fluid to conform the composite material to a punch with the desired part shape. This method is similar to sheet metal hydro-forming, which also uses pressurized fluid to conform a metal material to a punch of its final shape. The thermo hydro-forming for composites was developed and patented at Michigan State University’s Advanced Materials Manufacturing Laboratory.

The thermo hydro-forming has several advantages compared with match die molding. The heated and pressurized fluid can be used to keep the composite material at the forming temperature, which eliminates the need for heating the tool required in match die molding. Also, the use of fluid is similar to using a bottom die, which serves as a female die. So only the male die is required and there is a reduced cost for designing new tools and dies. Moreover, since fluid is used to form the part, the force vector that pushes the material to the punch is always normal to the material surface. The evenly distributed force along the material surface can keep uniform thickness of the material and reduce out-of-plane warping significantly. This hydro-forming method allows forming of deep drawn parts without the problem of developing non-uniform thickness along the material surface [7].
Chapter 2. Literature Review

Modeling Techniques

ABD matrix is the most common way to characterize laminated composite material in classical lamination theory (CLT) [8]. The ABD matrix has an assumption of plane stress, which makes the 9*9 stiffness matrix to be reduced to a 3*3 matrix. Then it is extended to a 6*6 matrix to account for the effects of extension-bending, and bending stiffness. The ABD matrix can be used to make good predictions for laminated composite material after having been formed. But it has limited accuracy for characterizing forming process of composite material, since it does not take changes of fiber orientation into account. Strains produced during the forming procedure can cause the fiber reinforcement to change its direction, which will change the effective properties of the overall composite material. Also, the model assumes a perfect bonding between laminates.

The fiber reinforced material can have different mechanical behavior, based on the type of reinforcement, orientation of the fibers, etc. In an ideal situation where fibers are truly randomly orientated, the composite material can be assumed to exhibit an isotropic material property. Zampaloni modeled a simulation of forming hemispherical cups from random continuous fiber reinforced polypropylene matrix composite material, with no prescribed directionality during the manufacturing process, using a commercial code MARC [9]. Only half of the sheet was analyzed by assuming symmetric behavior of the
overall part, and a rigid-plastic incremental analysis that used large displacements and an updated Lagrangian procedure was used for modeling the deformation process. It is found that although the material could theoretically show isotropic property, it is the actual manufacturing process that does impart some directionality to the material as it is being deformed. Also, by Environmental Scanning Electron Microscopy (ESEM) image taken of an undeformed glass mat fiber reinforced thermoplastic with a random orientation, it is seen that there seem to be some directionality, especially in x-direction. This makes the isotropy assumption invalid. Furthermore, fiber orientation after deformation in this model is not available.

Most of woven composite modeling assume homogeneous material properties that have been developed based on well-developed mathematical theory, which can reduce simulation time [10]. However, those models assume orthotropy of woven FRT throughout the whole deformation process. This means that the fiber directions in woven composite material stay perpendicular to each other, even after forming procedure. However, this assumption is not valid for analyzing forming of woven FRT, since fiber orientations change significantly during deformation. Then some efforts have been made to take fiber angle changes into account to the constitutive model, which leads to a non-orthogonal continuum constitutive equation.

For textile composites, Peng and Cao proposed a dual homogenization and finite element approach to characterize this kind of composite material [11]. The model uses a combined approach of homogenization method and finite element formulation to predict the effective nonlinear elastic moduli of textile composites. Since there are various kinds
of composite materials on the market, it is very time-consuming to obtain material properties by experiments, they use the homogenization method to study the material behavior of a single fiber yarn on a meso-microscopic level, and it is based on the properties of the constituent phases. Then a unit cell is built to enclose the characteristic periodic pattern in the textile composites, and the effective nonlinear mechanical stiffness tensor can be obtained numerically as functions of element strains.

Later, Xue, Peng and Cao propose a non-orthogonal constitutive model for characterizing woven composites [12]. During thermoforming of woven fabric reinforced composites, there usually will be large in-plane shear deformation, which induces additional anisotropy into the composite material. The model makes assumptions of incompressibility, the tensile and shear responses in the non-orthogonal material coordinates are decoupled, and the compressive stiffness is ignored. Based on stress and strain analysis in both the orthogonal and non-orthogonal coordinates, as well as the rigid body rotation matrices, the model can make a good prediction with respect to the experimental data. However, this model still needs to consider effects of temperature and viscous behavior of the resin.

Then, Cao et al. proposed a modeling approach to include the temperature effect in thermo-stamping of woven composite materials [13]. Since thermo-stamping process has relatively higher efficiency than autoclave stamping forming process, it is important to develop a model to predict mechanical behavior of the composite material during deformation. The model proposed in this paper is based on the non-orthogonal material model mentioned as above, but now it takes temperature effect into account. The model
uses two states of material properties for the simulation, i.e., a high temperature state and a low temperature state to approximate the contact status between the tooling and the composite blank. By incorporating temperature effect into the non-orthogonal material model, the equivalent material properties will be updated, which will affect the stamping simulation, consequently affecting the maximum draw depth of a composite blank. However, even this model has drawbacks. It purely relies on the assumption that all tooling is represented by analytical surfaces, and this is not true for complex geometries.

Autoclaves are often used for curing process of composite materials, and provides a controlled cycle of temperature and pressure in this process. However, upon removal from the tool, residual stresses will rise, and often it causes the deformed composite part to distort from its desired shape. Yuan et al. developed an analytical model on through-thickness stresses and warpage of composite laminates due to tool-part interaction [14]. By making the assumption that slip occurs between tooling and the composite material, and the knowledge of the stress distribution through thickness, the analytical model can predict through-thickness residual stresses and warpage for the composite material during the curing process in the autoclave, without extensive resin characterization.

**Material Characterization Methods**

Tests such as the three-point bending can be used to evaluate the properties of the entire specimen as a structure, so it is important to understand the mechanical response when there is an applied load to the composite structure. It will be a lot easier to understand homogeneous materials, since they respond isotropically when loaded in different orientations. However, when dealing with composite materials, the anisotropy created by
the orientations of reinforcing fibers for each ply has to be taken into account. Such kind of material response and structures needs an orthotropic material model coupled with an assumption of plane stress. The stiffness along the fiber axis and perpendicular to the fiber axis as well as the Poisson’s ratio and shear modulus will be taken into account in such a model. With those terms, a reduced compliance matrix can be obtained, which can be used to predict the mechanical response when a stress is present.

Zampaloni developed a material model which tracks preferred fiber orientations of randomly oriented, unidirectional or woven composite sheets [9]. At first, the model was developed to predict the response of randomly orientated fibers. For woven or unidirectional composites, it will be easy to determine which orientation will give the stiffest response. However, when dealing with composites with random fiber orientations, this cannot be done easily. By using a squeeze flow test, Zampaloni was able to determine which directions have the greatest amount of reinforcing fibers, and which orientations have the least. He was able to determine the orientations worth tracking in his material model just by assuming that orientations with the largest amount of reinforcing fibers would be the stiffest. Then, the material properties can be extracted when these orientations are determined.

Young’s modulus is typically the first property to be determined. This is standard characterization for composite materials based on ASTM D3039 [15]. In this standard, a thin strip of composite specimen with a constant rectangular cross section is mounted in a mechanical testing machine with a specialized grip to load the specimen in tension monotonically while recording the force. The stress-strain relationship of the material can
be determined through this process. And properties like ultimate tensile strain, tensile modulus of elasticity, Poisson’s ratio can be obtained.

Measurement of shear modulus is also necessary. Two methods are reviewed for obtaining shear modulus: the Iosipescu shear test [16] and V-notch rail shear test [17]. Both of these tests have a rectangular specimen with notches cut in order to concentrate shear stress at the neck. It is how the specimen is loaded that differentiates these two tests. The Iosipescu method utilizes a special equipment which loads the specimen along its edges by exerting compressive force. The V-Notched rail shear method uses another specialized fixture to grip the faces of the specimens and shears the specimen in tension. These two methods apply different forces in different locations of the composite specimen, therefore, it is reasonable to expect that each of the two approaches can give slightly different shear modulus. Yan-lei et al. evaluated these two methods in his study [18]. Although the Iosipescu shear specimens can give good results, he claims that edge crushing can be an issue due to the way the specimen is loaded into the fixture. And V-notched rail shear method can get rid of such unacceptable failure and it uses a larger gauge section, so this method has more advantages, and therefore it is preferred.

Thermo-stamping of composite sheets has been an efficient way to impart complex shapes into composites, since it only takes seconds to form the composite sheet into the desired shape. According to Liu et al., during the thermo-stamping process, shear is the main deformation mechanism, and the main parameter that is influencing woven fabric formability is the shear resistance [19]. A picture frame model was developed, using the kinematic and elastic analysis of the picture frame experiments. Then, the stress tensor can
be obtained from the proposed shear model, and by using Hooke’s law, shear properties of the woven fabrics can be calculated. An analytical solid mechanics model was also proposed to predict the shear properties, hoping to eliminate extensive experimental characterization. However, the analytical model had significant differences with experimental data. Because frictional resistance factor was not considered in the study.

Recently, Alshahrani and Hojjati proposed a new test method for characterizing the bending behavior of textile prepregs during forming simulation with both rate and temperature dependencies [20]. This new method is based on the vertical cantilever test, with metallic custom grips to clamp the sample vertically, while a linear actuator controls the sample deflection and applied rate. And a radiant heater is used to provide the sample with testing temperature, which is monitored by an infrared camera. By adjusting the testing speed from the actuator’s controller, the rate dependent effect can be measured. The testing results show that bending stiffness is about 20% higher in the warp direction than in the weft direction for satin woven carbon/epoxy prepregs. The results can be used for aiding future design. Moreover, the experimental results show that the bending behavior is rate-dependent, because of the viscoelastic behavior of the prepreg. However, the new testing method does not include the viscoelastic behavior at different temperatures. And the bending friction coupling effects might still have some influences during the forming simulation of laminated composites.
Chapter 3. Numerical Methods

Numerical simulations require a solid background on properties of laminated composites, since fiber directions, number of layers, lay up sequence, etc. can affect the laminate properties during deformation. The goal of developing numerical model is to produce a good agreement between simulations and experiment results, so that the results could be predicted before experiments.

Constitutive Model

Composite materials generally have elastic stress-strain behavior until failure. The stress-strain relation can be given as follows:

\[ \sigma_{ij} = C_{ijkl}\varepsilon_{kl} \] (1)

The stiffness tensor C has 81 independent terms, which are too many to use as a constitutive equation. Some assumptions have to be made. Tsai et al. showed that the stress, strain, compliance and stiffness matrices are symmetric, and the stress and strain are 2\(^{nd}\) order tensors, the number of independent terms is reduced from 81 to 36 [21]. Also, since the material stiffness is assumed to be symmetric, the number of independent terms in C is reduced to 21. Then another assumption is made that the material is exhibiting orthotropic behavior, which contains 3 planes of symmetry. Then equation (1) becomes
\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\sigma_3 \\
\sigma_4 \\
\sigma_5 \\
\sigma_6
\end{bmatrix} =
\begin{bmatrix}
C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\
C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\
C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\
0 & 0 & 0 & C_{44} & 0 & 0 \\
0 & 0 & 0 & 0 & C_{55} & 0 \\
0 & 0 & 0 & 0 & 0 & C_{66}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\varepsilon_3 \\
\varepsilon_4 \\
\varepsilon_5 \\
\varepsilon_6
\end{bmatrix}
\] (2)

Furthermore, individual layer will be modeled, which means that an assumption of plane stress can be made. Then equation (2) can be reduced to equation (3).

\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\sigma_6
\end{bmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\varepsilon_6
\end{bmatrix}
\] (3)

The reduced stiffness Q assumes that the material frame (1,2,3 direction) is aligned with its global coordinate (x, y, z axes). However, these coordinates do not have to align with each other, so a distinct notation is used to avoid confusion. The 1-direction from the material frame is always aligned with the fiber orientation, while the 2-direction is fiber’s transverse direction within the plane. The global coordinate is fixed in space, while material frame can move under loading. The notation can be shown in figure 5.

Figure 5. Material frame and global coordinate [7]
Stresses will be experienced in more than one direction within a plate, so Poisson’s ratio becomes important. The definition of Poisson’s ratio is the ratio of the strain perpendicular to the loading direction, to the strain along with the loading direction.

\[
\nu_{12} = \frac{\varepsilon_T}{\varepsilon_L} = \frac{\varepsilon_2}{\varepsilon_1} \quad \text{For loading along the fibers, or} \tag{4}
\]

\[
\nu_{21} = \frac{\varepsilon_L}{\varepsilon_T} = \frac{\varepsilon_1}{\varepsilon_2} \quad \text{For loading along the transverse direction} \tag{5}
\]

Then the strain component will decrease due to the contraction of Poisson’s effect caused by the force perpendicular to the applied force:

\[
\varepsilon_1 = \frac{\sigma_1}{E_1} - \nu_{21}\varepsilon_2 \quad \text{and} \quad \varepsilon_2 = \frac{\sigma_2}{E_2} - \nu_{12}\varepsilon_1 \tag{6}
\]

The shear force can be defined as:

\[
\tau = \gamma_{12}G_{12} \tag{7}
\]

And the above equations can be written in the matrix form as:

\[
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\gamma_{12}
\end{bmatrix} =
\begin{bmatrix}
S_{11} & S_{12} & 0 \\
S_{12} & S_{22} & 0 \\
0 & 0 & S_{66}
\end{bmatrix}
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{bmatrix} \tag{8}
\]

Where,

\[
S_{11} = \frac{1}{E_1} \quad S_{12} = -\frac{\nu_{12}}{E_1} = -\nu_{21} \quad S_{22} = \frac{1}{E_2} \quad S_{66} = \frac{1}{G_{12}} \tag{9}
\]

The S matrix is called compliance matrix, and by inverting the matrix, a reduced stiffness matrix, Q, can be shown as:

\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{bmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\gamma_{12}
\end{bmatrix} \tag{10}
\]
\[ Q_{11} = \frac{E_1}{1-\nu_{12}\nu_{21}}, \quad Q_{12} = \frac{\nu_{12}E_2}{1-\nu_{12}\nu_{21}} = -\frac{\nu_{21}E_1}{1-\nu_{12}\nu_{21}}, \quad Q_{22} = \frac{E_2}{1-\nu_{12}\nu_{21}}, \quad Q_{66} = G_{12} \] (11)

Often, the loading direction does not align with material direction, then the stresses and strains need to be transformed into the coordinates that do coincide with fiber directions. Then a transformation matrix, \([T]\), is defined as follow:

\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{bmatrix} =
\begin{bmatrix}
\cos^2\theta & \sin^2\theta & -2\sin\theta\cos\theta \\
\sin^2\theta & \cos^2\theta & 2\sin\theta\cos\theta \\
\sin\theta\cos\theta & -\sin\theta\cos\theta & (\cos^2\theta - \sin^2\theta)
\end{bmatrix}
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}
\] (12)

Also, the stiffness and compliance matrix after transformation can be defined as follow:

Stiffness matrix: \([\bar{C}] = [T][C][T]^T\) (13)

Compliance matrix: \([\bar{S}] = [\bar{C}]^{-1}\) (14)

**Obtaining Material Properties**

With the constitutive model developed, the stiffness matrix, \(Q\), can now be calculated. Four constants, \(E_{11}, E_{22}, E_{12}, G_{12}\), need to be obtained to construct the stiffness matrix.

\(E_{11}\) is the Young’s Modulus of unidirectional composite in its fiber direction. The most common way to obtain this constant is to use the rule of mixture as follow:

\[ E_{11} = E_{F1}V_F + E_{M1}(1 - V_F) \] (15)

Where \(E_F\) and \(E_m\) are the Young’s Modulus of fiber and matrix, respectively. And \(V_f\) represents the fiber volume fraction of the composite. It is reasonable to use \((1 - V_f)\) to represent the volume fraction of the matrix, since the composite is mainly composed of fiber and matrix, and the assumption that zero void exists, results in:
\[ V_f + V_m = 1 \] 

However, even among the highest quality of composite material, voids are inevitable. Therefore, the zero-void assumption is highly idealized. But usually, the percentage of voids are small enough to be ignored. Moreover, equation (16) assumes that there is a perfect bond between fiber and matrix components. And the following equation takes continuity of mass into account, and it is represented by Mohammed et al [22].

\[ V_f = \frac{v_{fo}}{\sin \alpha} \] 

Where \( v_{fo} \) is fiber volume fraction before deformation and \( \alpha \) is the angle between two orthogonal fibers, a-fiber and b-fiber, before deformation starts. In general, the assumption of zero void and perfect bond between fiber and matrix will introduce errors, but they are usually to be ignored because the error is too small.

\( E_{22} \) is the Young’s Modulus of unidirectional composite in its transverse direction. And the equation to determine this quantity is to invert the rule of mixture:

\[ E_{22} = \frac{E_{F2}E_{M2}}{E_{F2}V_M + E_{M2}V_F} \] 

Poisson’s ratio is also obtained by rule of mixture as below:

\[ v_{12} = v_MV_M + v_FV_F \] 

And

\[ v_{21} = \frac{E_{22}}{E_{11}} v_{12} \] 

Also the shear modulus can be defined in the rule of mixture:

\[ G_{12} = \frac{G_M G_F}{V_M G_F + V_F G_M} \]
Therefore, once the properties of fibers and matrix and volume fraction of fiber are obtained, all the constants mentioned above could be calculated accordingly. The stiffness matrix and compliance matrix of the laminated composite could be calculated, transformation matrix needs to be considered when the loading direction is not aligned with the principal material direction, so that the new stiffness and compliance matrix could be obtained for the laminated composite after rotation.

**Preferred Orientations**

The constitutive model mentioned above can be used to describe properties of unidirectional layup laminated composite. However, in real situation, fiber angle will change during deformation. Therefore, PFO model, which was developed by a former student, is used to trace the fiber angle change during deformation. Figure 6 represents a laminated composite composed of two layers of lamina. The stiffness of the overall material can be assumed to be the summation of the stiffness tensors of the two individual fiber orientations. Assume that one lamina, A-fiber, has 0-degree fiber orientation, and the other one, B-fiber has 90-degree fiber orientation. Before deformation, these fibers are orthogonal to each other. But during the process of deformation, these fibers will not always remain orthogonal to each other. The summation process can be illustrated in the following figure.
If the material cannot be fully characterized by two preferred orientations, additional fiber orientations could be added.

Global coordinate is needed for using the summation of each PFO. By using equation (10), the stiffness tensor can be obtained for the composite material. Then by using transformation matrix from equation (12), the stiffness tensor can be rotated to the global coordinate system. Next, the sum of stiffness for each PFO can be utilized when the stiffness tensor is rotated to the global coordinate, in order to get the overall properties of the material. It can be demonstrated as follows:
Laminated composite with two layers of fiber

The fiber orientation and its transverse direction for a-fiber and b-fiber and a global coordinate (red)

\[ \bar{Q}_{ij}^a = T(\beta)Q_{ij}T(\beta)^T \]
\[ \bar{Q}_{ij}^b = T(\alpha + \beta)Q_{ij}T(\alpha + \beta)^T \]

The stiffness is calculated for each PFO and then rotated to the global coordinate

\[ \bar{Q}_{ij} = \bar{Q}_{ij}^a + \bar{Q}_{ij}^b \]

The overall stiffness is calculated for a-fiber and b-fiber after rotation

Figure 7. Rotation of PFO to the global coordinate [7]
The process mentioned above is used to find the overall stiffness tensor of a composite material. And typically, the fiber orientations are specified by the user when creating simulation models, and it will make stiffness easy to calculate for the laminate.

However, as deformation proceeds, the angles of $\alpha$, and $\beta$ will change, and it will result in a change of transformation matrix. The overall properties of the laminated composite material will change due to the change of fiber angles. These changes are important, since they will determine how the composite material behave at each deformation increment. Take a small differential element with a-fiber and b-fiber with its global coordinate for an example. A deformation gradient tensor acts on the material.

$$F = RU$$  \hspace{1cm} (22)

Where $F$ is the deformation gradient tensor, and it can be decomposed into a product of two second-order tensors, which are $R$ the rotation tensor, and $U$ the right stretch tensor. The differential element is acted by the deformation gradient tensor by stretch and rotation tensors. The structure coordinate system is acted by the rotation tensor, $R$, so that the coordinate remains orthogonal after deformation. The fiber orientations are affected by the rotation and stretch tensor, which affect the angle of each fiber to the material coordinate system.
From the example, in the undeformed shape, the fibers are aligned with the SCS, and in the deformed shape, the SCS is changed by the rotation tensor R, and fiber angles are changed by the deformation tensor, which affects the stiffness matrix of the differential element, and the element will behave according to the stiffness tensor for the next step of deformation.

**Implementation into Abaqus**

The PFO model is implemented in Abaqus explicit via a user subroutine. The explicit solver allows the strain increment, and deformation gradient tensor to be fed into the subroutine for each progressive time step. Then the change of SCS and the fiber orientations during the current time step are determined by the user subroutine, using the deformation gradient tensor, as shown in figure (8). Next, the stiffness of the PFO can be obtained once those changes are determined. And the stiffness needs to be rotated to their material coordinate. These steps are repeated for every PFO. When all the PFO stiffness are rotated to their material frames, they are then summed up to get overall stiffness.
Multiply the strain increment with stiffness, the stress increment can be obtained for the time step. A flow chart of this process can be seen Figure 9.

Figure 9. Flow chart of PFO model implemented in Abaqus/CAE [9]
Analytical & Simulation Discussion

Some typical values of composite can be found in the table below.

<table>
<thead>
<tr>
<th>Fiber</th>
<th>$E_{f11}$</th>
<th>230 Gpa</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$E_{f22}$</td>
<td>230 Gpa</td>
</tr>
<tr>
<td></td>
<td>$\nu_{12}$</td>
<td>0.2</td>
</tr>
<tr>
<td></td>
<td>$G_{12}$</td>
<td>95.83 Gpa</td>
</tr>
<tr>
<td>Matrix</td>
<td>$E_m$</td>
<td>3.5 Gpa</td>
</tr>
<tr>
<td></td>
<td>$\nu_m$</td>
<td>0.4</td>
</tr>
<tr>
<td></td>
<td>$G_{12}$</td>
<td>1.25 Gpa</td>
</tr>
<tr>
<td>$V_f$</td>
<td></td>
<td>0.6</td>
</tr>
</tbody>
</table>

Where $E_{f11}$ and $E_{f22}$ are the Young’s modulus of fiber in its fiber direction, and transverse direction. $\nu_{12}$ and $G_{12}$ are the Poisson’s ratio and shear modulus of fiber. $E_m$, $\nu_m$ and $G_{12}$ are the Young’s modulus, Poisson’s ratio, and shear modulus of the matrix, respectively.
From the constitutive equations in the constitutive model section for calculating compliance and stiffness matrix for a single layer of a laminate after transformation, the new young’s modulus for fiber direction and its transverse direction can be obtained simply from $S_{11}$, and $S_{22}$ elements of the new compliance matrix. Also, shear modulus can be calculated from $S_{33}$ element from the new compliance matrix. Equations are shown below:

\[
E_{11} = \frac{1}{S_{11}} \tag{23}
\]

\[
E_{22} = \frac{1}{S_{22}} \tag{24}
\]

\[
G_{12} = \frac{1}{S_{33}} \tag{25}
\]

By implementing the logic into Matlab, graphs representing Young’s modulus and shear modulus can be shown in Figure 10. The fiber angle on x-axis means the angle between fiber and loading directions.

![Modulus plot](image)

Figure 10. Modulus plot for single layer
By putting 0 degree into the Matlab code, the Young’s modulus along the fiber direction, transverse direction and the shear modulus are shown in Figure 11 (unit: Gpa):

\[
\text{Finalresult} =
\begin{pmatrix}
139.4000 \\
8.5547 \\
3.0650
\end{pmatrix}
\]

Figure 11. Properties of a single layer with 0 degree

The results from Matlab are the basic material properties for Abaqus simulation later.

For multi-layer composite material, the new stiffness after transformation is calculated for each layer. Then all the stiffness matrices are added, and divided by the number of layers to get an average stiffness matrix. Since a lamina has the strongest strength along its fiber direction and least strength in its transverse direction, different stacking sequences can also affect the overall properties of the laminated composite material. Therefore, different stacking sequences can have different response during certain loading. For example, consider a laminated composite with 14 unidirectional layers. Three different stacking sequences are used to plot Young’s modulus and shear modulus. The different stacking sequences are [90/0/90/0/90/0/90]s, [90/45/0/90/45/0/90]s, [45/-45/45/-45/45/-45/45]s
Figure 12. Young's modulus plot in fiber direction

Figure 13. Young's modulus in transverse direction
The angle in the plots is the loading angle with respect to the global coordinate system. The plots give an understanding of how the material behaves under loading in different directions. And this helps to select appropriate stacking sequence for specific design requirements.
Numerical Validation

It is important to understand that during numerical simulation, the composite material will respond as the properties calculated from Matlab. The numerical simulation software used is Abaqus/CAE package.

Numerical Setup

The test used is a simple tension test. A composite material with a 500mm x 100mm dimensions is shown in Figure 15.

Figure 15. Dimension of a composite specimen
The part is created in shell planer base feature. Material properties are the results calculated from Matlab. And a shell composite section is assigned to the part. The fiber direction is along the x axis and its transverse direction is along the y axis, which are shown below.

Figure 16. Material properties of the specimen
A static general time step with a time period of 1 is created in the model. A standard four-node, reduced integration, S4R shell element is used with a total of 500 elements in the simulation. For boundary conditions, the bottom edge is set to YSYM and 50 mm displacement is applied to the top edge to stretch the specimen.
Figure 19. Bottom boundary condition of the specimen

Figure 20. Top boundary condition of the specimen
From the result, by summing all the reaction forces along the bottom edge nodes, and dividing the force by the bottom cross-sectional area, a stress vs. time curve can be obtained from the software. Also, by picking a node on the top edge of the specimen, a strain vs. time curve can be obtained. Combining these curves, a stress vs. strain curve can be created for the composite specimen. Young’s modulus can be demonstrated by calculating the slope from the stress vs. strain curve. The 2 different stacking sequences shown before are used for validation purposes.

For the stacking sequence [45/-45/45/-45/45/-45/45]s, the Young’s modulus calculated from Matlab is 11.3396 Gpa for both fiber and its transverse direction, and the stress vs. strain curve obtained from Abaqus is shown in Figure 22.
For the stacking sequence \([90/0/90/0/90/0/90]\), the Young’s modulus calculated from Matlab is 64.8745Gpa in the fiber direction, 83.6371Gpa in its transverse direction, and the stress vs. strain curve obtained from Abaqus is shown below.

Figure 22. Stress vs. strain curve for \([\text{-}45/45/\text{-}45/45/\text{-}45/45]\)

Figure 23. Stress vs. strain curve for \([90/0/90/0/90/0]s\) in its fiber direction
Comparing Matlab and Abaqus results, it can be concluded that the analytical approach and the numerical approach are in great agreement in regard to the values of Young’s modulus. This means that the behavior of the composite material in the numerical simulation follows the properties that are calculated from Matlab. Therefore, the numerical simulation results are reliable and can be used for more complicated simulation applications.

A simple shear test is performed in Abaqus/CAE to obtain shear modulus and compare it with that from Matlab code for the same stacking sequences from tension test. As it is shown below, the blue area is the deformed shape, and the grey area is the undeformed shape. The element has a 100mm x 100mm dimension, and the method used to calculate the shear modulus is also shown in Figure 26.

Figure 24. Stress vs. strain curve for [90/0/90/0/90/0/90]s in its transverse direction
Figure 25. Simple shear test

Figure 26. Calculation of shear modulus

\[ G = \frac{\tau_{xy}}{\gamma_{xy}} = \frac{F}{A} = \frac{Fl}{A\Delta x} \]  

(26)

Where \( \tau_{xy} = \frac{F}{A} \) = shear stress, F is the force, and A is the area on which the force acts, \( \gamma_{xy} \) = shear strain, \( \theta = \tan(\theta) = \frac{\Delta x}{l} \), since it is a small angle approximation. \( \Delta x \) is
the transverse displacement, and $l$ is the initial length. In equation (26), $l$ and $A$ are known from geometry of the element, and $\Delta x$ can be set in the displacement boundary condition of the simulation. Then $F$ can be obtained from summing up the reaction forces in the force direction.

Both stacking sequences have a $\Delta x = 1mm$ displacement to the right on top edge of the element. The bottom edge is fixed in all directions to prevent it from moving, and both vertical edges are fixed in $y$ direction so that there will not be any changes in the length. The cross-sectional area is $A = 280 mm^2$, and the initial length is $l=100mm$. For the stacking sequence $[45/-45/45\ldots45/-45/45]$, the shear modulus calculated from Matlab is 35.3894 Gpa. The force from Abaqus simulation result is 100.702 KN, and by applying the shear modulus calculation formula, $G_{12} = \frac{100.702 \text{ KN} \times 100mm}{280 mm^2 \times 1mm} = 35.965 \text{ Gpa}$.

For the stacking sequence $[90/0/90/0/90\ldots90]$, the shear modulus obtained from Matlab is 3.065 Gpa. The force from Abaqus result is 8.58473 KN, and from the shear modulus equation, $G_{12} = \frac{8.58473 \text{ KN} \times 100mm}{280 mm^2 \times 1mm} = 3.065975 \text{ Gpa}$. From the results, it can be seen that both shear modulus results are almost the same, the reason that there are slightly difference in numbers is that for $[45/-45/45\ldots-45/45/-45]$, the structure is undergoing simple shear deformation in Abaqus simulation, and Matlab calculations are based on pure shear deformation. Although, the shear strain is relatively small, still little difference could affect the reaction forces in numerical simulation, thus the shear modulus is affected. And Abaqus takes that into account in the simulation.

40
PFO Validation

Abaqus/CAE package assumes that the structural coordinate system does not change during deformation. This may be true with metals in some cases, however, the fiber orientation always changes under loading, which implies that the material axis also changes. The PFO model tracks alignment of the fiber during each time step, and then calculates the overall stiffness and updates material properties during the simulation.

To illustrate this point, a simple tension test model is developed in Abaqus/CAE package, and its result is compared with the PFO model for validation purposes. Figure 27 shows a 30mm x 30mm single square element subjected to a displacement of 15mm in the positive direction of the x-axis. The initial fiber orientation is set at 45 degrees (0.7854 radian) from the x-axis. The results are shown in Figure 27.

![Figure 27. Fiber orientation after deformation of the tension test](image-url)
The 1-axis, which is in blue color, is the fiber orientation after deformation, and 2-axis, which is in yellow color, is the transverse direction to the fiber. From the result, the fiber orientation remains at 45 degrees from the x-axis after the deformation. This is not a real situation for composite materials under loading, since fibers are free to move during the deformation. That means the fiber angle will not stay at 45 degrees after the deformation.

The same test with PFO subroutine is performed to demonstrate the fiber direction change after the deformation. The result is shown in Figure 28. SDV5 shows the updated fiber angle after deformation, unit is in degrees.

![Figure 28. Tension test result of fiber angle (degrees) after deformation with PFO model](image)

The PFO tracks the fiber orientation after deformation, which is shown as SDV5 in Figure 28. From the contour of SDV5, it can be seen that the fiber orientation changes from
45 degrees to 31.33 degrees. To validate this angle change, an analytical calculation is performed based on the geometry of the deformed shape. From Abaqus, the coordinates of the deformed element are shown in Figure 29.

![Figure 29. Coordinates of deformed element with PFO model](image)

The angle \( \theta \) is the new fiber orientation that needs to be solved. Simply taking \( \tan^{-1} \left( \frac{27.39714}{45} \right) \), the new fiber angle comes out to be 31.33 degree, which is exactly the same as the one obtained from the PFO simulation. The reason that fiber is always diagonally connected is that the PFO model always assumes a perfect bonding in the lamina.

A shear test was also performed to validate the PFO model. The same 30mm x 30mm single element was subjected to a displacement of 5mm on top of the square, and the initial fiber angle was assumed to be 45 degrees from the x-axis. The result is shown in Figure 30.
From the result, the fiber orientation that the PFO model tracks after the deformation is 44.59 degree in SDV5. An analytical calculation is also performed to prove that this angle is correct. A schematic of the calculation is shown in Figure 31.

From the PFO model’s subroutine, the rotation of the structural coordinate system can be obtained, which is 4.834 degree as shown in Figure 31. The red coordinate is the
structural coordinate system after the deformation, and the blue dashed line is the global coordinate. From the Abaqus software, the coordinate of the top right point can be obtained as (35, 29.1153), as shown. By taking the arctangent of this coordinate, a 39.756 degree can be calculated, which is the angle of the black dash line along the diagonal. Then by adding these two angles together, a new fiber orientation can be calculated after the deformation as: $39.756 + 4.834 = 44.59$, which is exactly the same as the angle that the PFO simulation predicted in Figure 30. The shear test also proves that the PFO model uses deformation tensor to update the overall properties of the composite material, which is subjected to both rotation and stretch tensors.

**Validation with experiments**

A uniaxial extension test is simulated to compare with experimental data obtained from references for the PFO model. This time the tension test is loaded in the vertical direction, and the positive y-axis is the loading direction. And the composite material is assumed to be a woven fabric preform with 45/-45 for its preferred fiber orientations. The composite material is a rectangular shape with 203.2mm x 406.4mm in dimension, as shown in Figure 32. For boundary conditions, the bottom edge is constrained such that all nodes are fixed and cannot move in any direction, and the top edge is subjected to a 101.6 mm displacement in the positive y-axis direction, to create a 25% elongation. The simulated result is shown in Figure 33. The SDV5 parameter shows the updated fiber angles (in unit of degrees) after the deformation.
From the simulation result, three distinct deformation areas can be identified with different fiber angle changes. It can be seen that there is no significant fiber angle change.
in the blue areas on top and bottom parts, as the fiber angle after deformation is around 47.42 degrees compared to 45 degrees for its initial orientation. Also, there are relatively large fiber angle changes in the middle area of the specimen, and a little less angle change on the side areas. It is interesting to note that there are different modes of deformations taking place in the stretched woven sheet. Sidhu et al. also identified three different modes of deformation in their experiment, as shown in Figures 34 and 35 [23].

Figure 34. Deformed preform at elongation of 25 %
Figure 35. Three different modes of deformation in experiment

From the experimental results, it can be seen that there are no significant fiber angle changes in Zone I. There are both shearing and sliding between yarns and some fiber angle change in Zone II. And there is mainly shearing deformation in Zone III. And the simulation results using PFO model is in good agreement with these experimental results. That implies that the PFO model’s prediction of the fiber angle rotation is in close agreement with the real deformation of a composite material subjected to a uniaxial tension deformation. Furthermore, this will also imply that the PFO model will be able to correctly predict the updated properties of the composite material after each incremental deformation. Given that the Abaqus/CAE model does not take into account fiber rotation after deformation, its prediction of material properties will not be correct either.

Next, simulation of stamping of a woven composite material with initial 0/90 degree preferred fiber orientation to make a hat section was also performed with the PFO model. The shape and dimensions of the tooling used in this stamping simulation is shown in Figure 36.
The shape of the lower die is similar to the upper die, with a slightly bigger dimension to compensate for the thickness of the composite sheet. The dimension of the sheet is 180mm x 280mm, with general contact property for interaction between the sheet and the dies. The assembled tooling and the composite sheet are shown in Figure 37. Figures 38 through 41 show the simulation results for woven composite sheets with 0/90 and 45/-45 initial fiber orientations, as well as experimental results. The SDV5 parameter shows the updated fiber angles (in unit of degrees) after the deformation.
Figure 37. Assembly of the stamping simulation

Figure 38. Stress distribution of the sheet (0/90 degrees)
Figure 39. Result of stamping process of the sheet (0/90 degrees), SDV5 is the updated fiber angle after deformation in degrees
Figure 40. Stamped hat section from experiment (0/90 degrees)
Figure 40 continued

Figure 41. Stamping simulation with 45/-45 PFO. SDV5 represents fiber angle orientations after the deformation in degree
From the finite element simulation results shown in Figure 39, it can be seen that for 0/90 PFO, there is no fiber angle change in most of the specimen during the deformation. There is very little fiber angle change in corner areas, where bending stresses are the highest. When compared with the actual samples, shown in Figure 40, it can be seen that there were no obvious fiber angle changes after stamping. This is probably due to the simple shape of the tooling, and that the stamping process does not stretch the sheet off-diagonally. In other words, there are not much shear deformation imposed in the composite sheet to cause fiber rotations. However, when the PFO of the composite sheet was changed from 0/90 to 45/-45 degrees, more fiber angles changed direction during the stamping process. As can be seen from Figure 41, although the simulation results show no significant angle changes after stamping, still the change in the fiber orientation around corner areas is slightly larger than that from 0/90 PFO simulation. Again, this shows the importance of tracking fiber alignment in the PFO model, which is neglected in the Abaqus/CAE model.
Conclusions

Throughout the course of this study, composite material has been studied for its types of reinforcements, its unique directionality properties, etc. When forming a composite material, it is necessary to understand the constitutive relations between stress and strain, so that the material properties such as Young’s modulus and shear modulus can be obtained from compliance and stiffness matrices. A former student, Zampaloni, developed a PFO model which tracks the fiber alignment of the reinforcing fibers within the material during the deformation. During the FEA simulation, each element is subjected to a unique deformation gradient tensor, which not only deforms the element but also changes the angle of PFO. Then, the material properties are updated according to the change in PFO. The PFO model has been used to simulate the simple tension test and shear test with Abaqus software. The results from these simulations indicate that the PFO model works well for predicting fiber orientation change during the deformation, which Abaqus/CAE model does not account for. Neglecting to update fiber orientations results in inaccuracies in the shape of deformed part as well as its material properties. Furthermore, the PFO model was used to simulate the uniaxial tension test and stamping of a hat section, and numerical results were compared with those obtained from similar experiments. The comparisons showed that the PFO model can track the fiber orientation correctly, and provide more precise results, which were also very close to the real forming situation.
Although, Abaqus/CAE model is computationally faster in obtaining preliminary results for composite simulation, the PFO model implemented into Abaqus as a user material subroutine (VUMAT), provides more reliable results. A shortcoming of the PFO model is the assumption of perfect bonding between layers, which can lead to wrong predictions when out of plane warping occurs. In the future, the PFO model needs to be updated in order to relax this assumption for forming simulations.
References


