CONSTRUCTION AND IMPLEMENTATION OF A MULTI-MATERIAL FINITE ELEMENT FOR THE STIFFNESS TENSOR PREDICTION OF A PLAIN WEAVE COMPOSITE LAMINA

Christopher Boise, David A. Jack, and Douglas E. Smith
Department of Mechanical Engineering
Baylor University

Abstract

As woven fabric composites become more popular in the aerospace and automotive industries, it becomes important to understand how various fiber reinforced laminated composites react to structural loadings. This paper presents a method to obtain the effective stiffness tensor of a woven fiber composite lamina through finite element analysis (FEA) of a representative volume element (RVE) through the use of a novel approach that allows individual finite elements to contain multiple materials. Typical meshing within the RVE is complicated by the undulation of the fiber tows within the RVE, and this paper introduces a unique formulation of two multi-material finite elements (MMEs) that allows the meshing to be performed independent of the woven geometry within the RVE. The methods introduced are formed by constructing a stiffness based correction factor to the multi-point Gauss integration during the formulation of the elemental stiffness. The results presented in this work demonstrate the methods for a woven fabric geometry similar to that found in many glass and carbon fiber laminates. Preliminary results for the stiffness tensor components show very good agreement with results obtained doing the full geometry dependent analysis using a commercial software package. Results for the material stiffness tensor components and the associated stiffness coefficients (i.e., Young's moduli, Shear moduli, etc.) are presented to compare the stiffness tensor components from the proposed MMEs against published experimental results, the full finite element model, and alternative material independent elements. In all cases, the proposed method is either better than or equal to alternative material independent elements.

Background

Woven fiber composites are becoming more popular in the automotive and aerospace industries today. Because of their height stiffness-to-weight ratio, they are being increasingly used in structural applications to replace heavier metals, often with the consideration being an improvement in the fuel efficiency. Therefore, there is an increasing desire to predict the stiffness and failure properties of these heterogeneous woven composite structures for design purposes.

One means of performing this analysis is through the use of analytic methods, where woven properties of a composite is described through the use of geometric functions. Classical laminate theory (CLT) provides a foundation on which many analytic micromechanical models are based upon (see e.g., [1] for a presentation of CLT). Ishikawa and Chou [2] developed a mosaic model based on simplified woven geometries for a material response caused by loading in one direction. Naik and Shemekar [3] improved upon the work of Ishikawa and Chou by accounting for the fiber structure perpendicular to the loading direction. They developed a closed-form analytic method that takes into account fiber undulation in both the fill and warp directions. Scida, et al. [4, 5], extended the work of Naik and Ganesh in a model called MESOTEX that can be used to study composites with weave types other than plain weave (i.e., twill and satin weave) composites. More recently, analytic micromechanical models, such as that presented by Zuo and Xie [6], have been used to optimize the effective stiffness of woven composites through the alteration of a
number of design parameters. Crookston, et al. [7], or Dixit and Mali [8] both provide excellent discussions on the current state of the art in the area of micromechanics predictions for woven composites.

Because of some of the underlying assumptions, such as geometry accuracy and assumed forms of the stress and/or strain field, these methods have fallen out of favor for use in stiffness property prediction. Several recent research efforts have transitioned over to numerical techniques such as the finite element method [7]. Using a set of periodic boundary conditions (such as those found in [9]), a representative volume element (RVE) can be analyzed and the overall stiffness tensor of the heterogeneous structure can be determined. Work has been performed to accurately model the geometry of woven fabric RVEs (see e.g. [10]). However, due to the complex geometry of a woven composite (i.e., the shape of the strand, fiber gap, resin pockets, etc.), a finite element model can become difficult to mesh, especially when considering a changing geometry (see e.g., [11]). Historically, to simplify the computation process, the modeled geometry had to be simplified. The impact of these simplifications on the results would then have to be determined separately (see e.g. [12]). Further, in the case of ultrasound scans [13] or micro-CT scans [14], the geometry of a composite is only known pointwise, as opposed to continuously. It would be ideal if analysis of these scans could be performed without having to explicitly rebuild a geometry from these pointwise data.

Traditionally, when a finite element model is meshed, the elements have to adhere to material boundaries. Whitcomb and Woo [15] take a preexisting mesh and combine multiple elements into macroelements and mathematically relate the unwanted degrees of freedom to the nodes that are desired to be kept. As an alternative to this type of approach, Zeng, et al., proposed a material independent element for use in predicting the stiffness [16] and strength [17] properties of three-dimensional braided composites. This element applies material properties at the numerical integration points, rather than within the element as a whole, to good effect; thus allowing multiple materials to be taken into account within one element without having to first build a traditional mesh. Caselman [18], in his thesis, improved on the accuracy of the work of Zeng by accounting for the differences in strain caused by the varying stiffness of each material. Through the use of a one-dimensional spring model, Caselman developed strain correction factors that were applied to the otherwise linear interpolation functions. Caselman’s method was applied to the analysis of short-fiber filled composites composed of isotropic constituent materials.

This paper will introduce two newly constructed multi-material elements (MMEs) to improve upon the mathematical form suggested by Caselman [18]. The form of these elements is fully explained by the author in the thesis [19]. The key improvements from these new approaches is the ability of individual elements containing more than two anisotropic materials within the same element. This proposed element is constructed such that it can entirely disregard material boundaries by having effective material properties applied at the integration points rather than over the element as a whole. Figure 1 shows a comparison between a traditional mesh and a mesh composed of multi-material elements. The goal of these material-independent elements is to simplify the meshing process so that more realistic and complicated woven geometries can be properly analyzed for their material properties without having to explicitly model the geometry. The results of the predicted stiffness of a plain weave fiberglass lamina are compared to full finite element simulations and those results predicted by methods constructed by previous authors in this area. In all cases investigated, both in this paper and in [19], the new MMEs yield an accuracy equal to or greater than all previous geometry independent meshing schemes.
Figure 1: The purpose of the material independent element is to convert a complicated mesh, such as the one on the left, to a simpler mesh that can ignore material boundaries and therefore provide simpler analysis, such as the mesh on the right.

## Finite Element Principles

### Derivation of the Weak Form

The finite element method is a means of numerically approximating the solution of a partial differential equation over a domain by recasting the system as a set of coupled linear algebraic equations. In the present study we begin from the constitutive equation based on Hooke’s Law (see e.g., [20]) expressed in three dimensions as

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

where repeated indices imply summation in three space. The finite element method divides the full domain into smaller subdomains called elements and estimates the solution of the differential equation expressed in Equation (1) over each of those elements through the use of interpolation functions. The approach combines the individual solutions into a large system of solutions that must be solved simultaneously. An intermediate step is the construction is the formation of the elemental stiffness matrix \( K^e \) subjected to elemental loads \( P^e \) that describes, for a structural system, the elemental displacements \( u^e \) as (see e.g. [21])

\[
K^e u^e = P^e
\]  

The displacements or the forces are assumed continuous between element boundaries, and can be formed into a global system of linear algebraic equations in terms of the nodal displacements and nodal forces. Reddy [19] along with many other authors provide excellent discussions on the formulation of these global systems. Regardless, the elemental stiffness matrix is derived from the base partial differential equation being solved, and in the case of Equation (1) for structural mechanics the elemental stiffness matrix can be expressed as:

\[
K^e = \int_{\Omega^e} B^T C B \, d\Omega^e
\]  

where \( C \) is the contracted 6 \times 6 form (see e.g., [1]) of the material stiffness tensor described in Equation (1) and \( B \) is the strain displacement matrix, which contains derivatives of the \( N \) interpolation functions \( \psi_j^m \) that describe the geometry of the element. The strain displacement matrix can be expressed by horizontally concatenating the 6 \times 3 matrix \( B_m \) for each of the \( N \)
interpolation functions as (where a comma implies a derivative in \( x_i \))

\[
\mathbf{B} = [\mathbf{B}_1 | \mathbf{B}_2 | \cdots | \mathbf{B}_N], \quad \text{where } \mathbf{B}_m = \begin{bmatrix}
\psi_{11}^m & 0 & 0 & 0 & \psi_{13}^m & \psi_{12}^m \\
0 & \psi_{22}^m & 0 & \psi_{23}^m & 0 & \psi_{21}^m \\
0 & 0 & \psi_{33}^m & \psi_{32}^m & \psi_{31}^m & 0
\end{bmatrix}^\top
\]

(4)

In standard finite element practice, elements are often mapped back to a “master” element via the Jacobian, \( \mathbf{J} \). This allows elements of unequal size and shape to be recast and evaluated using the same base geometry. More importantly, it allows a systematic approach that can be implemented within a looping structure for evaluating the integral of Equation (3) independent of the elemental geometry. In the present work, we employ Gauss quadrature, which evaluates the integral as the sum of the integrand evaluated at specific Gauss points \( \xi_i \) and multiplied by the appropriate weight, \( w_i \). Thus, in three dimensional systems the volume integral of Equation (3) becomes a triple sum.

\[
\mathbf{K}^e = \sum_{m=1}^{N_g} \sum_{n=1}^{N_g} \sum_{p=1}^{N_g} \mathbf{B}(\xi_1^m, \xi_2^m, \xi_3^m)^\top \mathbf{C}(\xi_1^n, \xi_2^n, \xi_3^n) \mathbf{B}(\xi_1^p, \xi_2^p, \xi_3^p) \det(\mathbf{J}(\xi_1^m, \xi_2^m, \xi_3^m)) \psi_{11}^m \psi_{22}^n \psi_{33}^p
\]

(5)

Notice that the above formulation allows the strain displacement matrix, the Jacobian, and of specific note, the stiffness matrix to be spatial variables within the element. Direct implementation of (5) will be shown in the following section to be inadequate, as was suggested by Caselman [18], but through appropriate modifications of the strain displacement matrix accurate solutions will be demonstrated.

**Material Property Prediction – Representative Volume Element (RVE)**

The representative volume element (RVE) is a geometry that represents one unit of an overall periodic geometry, such as is the case of the weave geometry within a composite lamina. The RVE is placed at any location with the domain and is able to be repeated infinitely many times within the domain without any overlap or gaps between individual RVEs. The most essential feature of an RVE in terms of boundary conditions is that the must remain no overlaps or gaps between RVEs under deformation. Xia, et al. [9], demonstrated that fixing one boundary and placing a displacement on the opposite boundary overconstrains the RVE and results in a solution for the RVE that is not able to be repeated without gaps or overlap.

Xia et al. [9] demonstrated that periodic boundary conditions do satisfy the appropriate boundary constraints, and may be applicable for predictions of the effective material stiffness tensor. Periodic boundary conditions apply displacements relative to the opposing face rather than constraining one face and displacing the other. By using periodic boundary conditions, displacements are applied at the nodes without constraining the entire boundary, allowing the boundary as a whole to freely displace. The periodic boundary conditions, as presented in Xia et al., can be stated as

\[
u_{ij}^{dest} = u_{ij}^{src} + c_{ij}
\]

(6)

where \( u_{ij}^{src} \) represents the source displacement on the \( i \) th face in the \( j \) th, direction, \( u_{ij}^{dest} \) represents the corresponding node that is being related to the source node, and \( c_{ij} \) represents the displacement obtained from the desired strain over the material element. This displacement for small strains can be expressed in terms of a cubical RVE geometry of dimensions \( L_1 \times L_2 \times L_3 \) as
\[ c_{ij} = \varepsilon_{ij} L_i \]  

Equation (7) can be imposed to apply various displacements upon an element, and the resulting stress state may be obtained. The combination of the applied strains and the resulting stresses are then used to obtain the 21 independent constants of the stiffness tensor expressed in Equation (1), assuming a fully anisotropic material. For each loading condition there will be 6 strains applied with 6 resulting stresses, therefore 21 multiple studies need not be performed to fully define the reduced stiffness tensor of the heterogeneous material. Table I shows one possible example of six sets of periodic boundary conditions (one for each strain state) that may be applied to fully resolve a fully anisotropic stiffness tensor. This particular set of values for \( c_{ij} \) generates a unit strain in each direction, which will end up simplifying the post-processing calculations.

<table>
<thead>
<tr>
<th>( j )</th>
<th>( c_{1j} )</th>
<th>( c_{2j} )</th>
<th>( c_{3j} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \varepsilon_{11} )</td>
<td>( L_1 )</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>( \varepsilon_{22} )</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>( \varepsilon_{33} )</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>( \varepsilon_{23} )</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>( \varepsilon_{13} )</td>
<td>0</td>
<td>0</td>
<td>0.5 ( L_1 )</td>
</tr>
<tr>
<td>( \varepsilon_{12} )</td>
<td>0</td>
<td>0.5 ( L_1 )</td>
<td>0</td>
</tr>
</tbody>
</table>

To obtain the effective stiffness, the volume average of each of the six stress values \( \bar{\sigma}_{ij} \) is computed at the end of each study as [9]

\[
\langle \bar{\sigma}_{ij} \rangle = \frac{1}{V} \iiint_V \sigma_{ij} \, dV
\]

Because each of the boundary conditions applied represent one individual strain state, each set of values of the volume averaged stress can be directly related to one column of the reduced stiffness tensor. For each column \( m \) of the reduced stiffness tensor, the relation becomes (no sum on \( i, j \))

\[
[C]_m = \left[ \frac{L_i \sigma_{ij}}{c_{ij}} \right]_m
\]

**The Multi-Material Element (MME)**

There are a myriad of reasons to develop an element that can cross material boundaries. In the present context of composites, the complexity of either generating a 3D solid models from experimental observations or of meshing a 3D solid model is often quite prohibitive.

The power of the concept of a multi-material element (MME) is shown in the flowchart of Figure 2. When using traditional finite elements, the mesh is built from the geometry; this is because each element is homogenous, so the interior material boundaries need to be known. This technique can be cost prohibitive to build for a number of cases. For example, in the case of examining progressive damage (see e.g., [11]), the geometry has to be rebuilt after each iteration as damage occurs in the part. This means, with traditional elements, a new mesh has to be generated. With MMEs, the same simple mesh can be used and the geometric changes can
instead be accounted for in the calculation of the global stiffness matrix at each iteration.

\[
\text{TRADITIONAL FINITE ELEMENTS}
\]

Geometry → Mesh → Solver → Solution

\[
\text{MULTI-MATERIAL ELEMENTS}
\]

Mesh → Solver → Solution → Geometry

**Figure 2:** A comparison of the steps required to solve a finite element problem using traditional elements versus MMEs. For MMEs, the geometry is considered in calculating the stiffness matrix of each element; therefore, a mesh can be generated independently of the geometry.

Another example where MMEs are useful is in the analysis of pointwise-defined geometries. Such geometries can be generated through ultrasound scans [13] or micro-CT scans [14]. While a geometry could theoretically be modeled with such data, it would be more convenient to instead use the pointwise-data directly without having to develop a scheme to convert the data into a model that could be easily meshed and analyze in a standard finite elements package.

With these applications in mind, the desired characteristics of the (MME) are to:

- Accurately predict the stiffness tensor of heterogeneous structures with complex geometries.
- Evaluate an element domain that contains more than one material type with an infinite gradient in properties along an internal boundary, such as between the resin and the fiber tow.
- Mesh a domain entirely independent of internal geometry. In other words, the mesh can be generated before internal geometry is considered.

There are several approaches, both within the literature and enhancements investigated as part of this research that are investigated in this paper and fully explored in the companion work [17] that satisfy the above metrics. The four approaches investigated are given below and are listed in order of increasing complexity and, as the results will demonstrate, in order of increasing accuracy.

**The Average Stiffness Element (ASE)**

The simplest of the MMEs is derived through a direct numerical evaluation of the average stiffness over an element; therefore, it is termed the Average Stiffness Element (ASE). The concept is that the material stiffness matrix $C$ can be estimated as the element-wise average of the individual material stiffness matrices $\bar{C}(x_1, x_2, x_3)$ over the volume of the element as
\[ \langle C_{ijkl} \rangle = \frac{1}{V} \iiint_V \bar{C}_{ijkl}(x_1, x_2, x_3) dV \]  \hspace{1cm} (10)

Equation (12) is simply the average value of the stiffness tensor over the element. This method is neglects any cross coupling terms (i.e., extension-extension as caused by the Poisson effect, shear-extension, shear-shear, etc.).

**Basic Multi-Material Element (B-MME)**

Zeng, *et al.* [16], devised a simple method to hybridize elements for use in mechanical and failure analyses of three-dimensional braided composites. Their approach is for a different system and purpose than that studied in the present context, the results from their method are notable. The method is a numerical technique that applies the material properties individually at each Gauss point \((\hat{x}_m, \hat{x}_n, \hat{x}_p)\) within the numerical integral calculation of the element stiffness matrix.

\[
K^e = \sum_{m=1}^{N_{gp}} \sum_{n=1}^{N_{gp}} \sum_{p=1}^{N_{gp}} B^T(C_{11}^{m}, \xi_1^{m}, \xi_2^{m}, \xi_3^{m}) B(C_{11}^{n}, \xi_1^{n}, \xi_2^{n}, \xi_3^{n}) \det(J) w_m w_n w_p \]  \hspace{1cm} (11)

This approach alleviates issues involved with meshing complex geometries by instead allowing multiple material properties to exist within the element. The element can refer to a geometric function, results from a finite element calculation of the element stiffness matrix.

**Tensile Modulus Corrected Multi-Material Element (TMC-MME)**

Caselman [18], in his thesis, demonstrated that the B-MME did not account for the change in strain that occurs when the material boundary is crossed within the element. This discontinuity in the true strain cannot be captured by the choice of continuous interpolation functions across the entire heterogeneous element.

Using the analogy of two one-dimensional springs in series, Caselman proposed a means to correct the longitudinal strain within a two-material element by using the material properties to adjust the derivatives through a series of correction factors on the interpolation functions (which are analogous to the strain in the finite element equations). He then expanded this method into three dimensions, thereby arguing that each line of Gauss points along a coordinate axis could be treated as a linear spring system.

Caselman’s original element, having been derived for short-fiber filled composites, did not consider orthotropic materials within the element, only isotropic materials. Further, there was no method provided to account for more than two sets of material properties within the element. Therefore, the authors have expanded upon Caselman’s original derivation; the modified element is termed the Tensile Modulus Corrected Multi-Material Element (TMC-MME).

The correction factors \(\alpha_i^q\) are based on the effective tensile modulus \(E_i^{eff}\) of each line of Gauss points stemming from a Gauss point \((\xi_1^m, \xi_2^m, \xi_3^m)\) in each of the orthogonal \(x_i\) directions. The effective tensile modulus is calculated using the linear volume fraction \(\beta_i^q\) of each material. The correction factors are calculated as (no sum on \(i\))
Caselman’s original correction factors proved to be reliable in predicting the material stiffness properties of short-fiber filled composites for various volume fractions with one significant limitation. In his preliminary volume fraction studies using two-elements, the transverse modulus $G_{23}$ tended to be under-predicted and the shear, shear-extension, and shear-shear coupling terms of the stiffness tensor are suspect due to the construction of the correction factors to be exclusive functions of the material stiffness.

Stiffness Tensor Corrected Multi-Material Element (STC-MME)

In this paper we seek to further extend the work of the TMC-MME to improve the results for non-isotropic materials, which is of unique interest to the woven laminate composite community. The full details of the derivation and justification of the extended element, termed the Stiffness Tensor Corrected Multi-Material Element (STC-MME), are provided in the thesis by Boise [19]. This approach does not use the engineering properties such as tensile modulus $E$, but instead uses the components of the stiffness tensor $C_{ijkl}$ from Equation (1) to correct the strain at each Gauss point. The motivation to use the stiffness tensor is that it directly relates stress to strain for three-dimensional problems.

The strain correction factors maintain a similar form as in Equations (12)-(14), with the stiffness tensor components instead of the Young’s moduli. For $i,j = \{1,2,3\}$ with no sum on $i,j$ the correction factors are expressed as (see [19] for full derivation)

$$
a_{ij} = \frac{c_{ij}^{\text{eff}}}{c_{ij}^{q}}
$$

(16)

where the correction factors are spatial variables as the local stiffness $C_{ijkl}$ will potentially vary within the element. There are two methods that have been investigated for evaluating the effective stiffness $C_{ijkl}^{\text{eff}}$ within the element, depending upon whether $c_{ij}^{\text{eff}}$ is for axial loads or shear loads. For the corrections in the strain-displacement matrix expressed in terms of the axial strains ($\alpha_{ii}$, no sum on $i$), the effective stiffness used in Equation (16) becomes (for $i = \{1,2,3\}$, no sum on $i$)
\[
\frac{1}{C_{\text{eff}}} = \sum_{q=1}^{N_{\text{gp}}} \beta^q_i \frac{C^q_{i \text{eff}}}{C^q_{i ii}}
\]

Figure 3 demonstrates visually which properties are needed to calculate the effective stiffness at the marked point (the black “x”) in each of the axial directions. Intuitively, if each line of Gauss points is treated like a set of springs in series, then only the material properties extending orthogonally from the marked point are needed for the corrections at the marked point. For example, only the \( C_{3333} \) component at each point along the line passing through the marked point (including the property at the point itself) in the \( x_3 \) direction (marked in blue) is needed to calculate \( C^\text{eff}_{3333} \).

Figure 3: For the marked point (the black “x”), only the properties passing through the point in the \( x_3 \) direction (distinguished with a blue line) are needed to calculate \( C^\text{eff}_{3333} \). Similar argument can be made for \( C^\text{eff}_{1111} \) (red) and \( C^\text{eff}_{2222} \) (green).

The effective stiffnesses for the corrections related to the shear strains \( (\alpha_{ij}, i \neq j) \) require contributions from the two directions \( x_i \) and \( x_j \) because the shear strain is applied over an area. To account for this bidirectionality, the properties from the \( q^\text{th} \) material in both directions are first averaged together to calculate \( \overline{C^q_{ijij}} \). This value is then used in calculation of the effective property \( C^\text{eff}_{ijij} \). This is mathematically expressed as (noting that \( q_i \) refers to the \( q^\text{th} \) material in the \( x_i \) direction, and with no sum on \( i,j \)):

\[
\frac{1}{C_{\text{eff}}} = \sum_{q=1}^{N_{\text{gp}}} \beta^q_i \frac{C^q_{i \text{eff}}}{C^q_{i ii}}
\]

\[
C^q_{ijij} = \frac{1}{2} (C^q_{ijij} + C^q_{ijji})
\]

\[
\beta^q_i = \frac{1}{2} (\beta^q_i + \beta^q_j)
\]

Figure 4 provides a visual on how properties are combined to calculate \( C^q_{2323} \) for each \( q \) ther in the summation in Equation (18). For the marked point (the black “x”), the first \( C_{2323} \) term on the \( x_2 \) line is averaged with the first \( C_{2323} \) term on the \( x_3 \) line to calculate \( \overline{C^q_{ijij}} \). This pattern continues for each value of \( q \).
Figure 4: For the marked point (the black "x"), \( C_{2323} \) in the \( x_2 \) direction (green) and the \( x_3 \) direction (blue) are needed to calculate \( C_{2323}^{\text{eff}} \). These properties are averaged at each \( q \) to calculated \( \overline{C}_{2323}^q \).

Once each of the corrections \( a_{ij}^n \) of Equation (16) are calculated at a Gauss point, they are inserted to the \( \mathbf{B}_m \) matrix of Equation (15) in a similar fashion as Caselman. Then the numerical integration of Equation (5) may be evaluated at each Gauss point to obtain the corrected elemental stiffness for an element containing multiple materials. Also note that this method allows for fully anisotropic materials whereas the previous hybrid element required the isotropic young's moduli, which is undefined for an anisotropic material. Table II summarizes the individual calculations of \( \alpha_{ij}, C_{ijij}^{\text{eff}}, C_{ijij}^q, \) and \( \beta_{ij}^q \) for the presented study. The derivation of these terms are provided in Boise [17].

<table>
<thead>
<tr>
<th>( i )</th>
<th>( j )</th>
<th>( \alpha_{ij} )</th>
<th>( \left( C_{ijij}^{\text{eff}} \right)^{-1} )</th>
<th>( \overline{C}_{ijij}^q )</th>
<th>( \beta_{ij}^q )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>( \frac{C_{1111}^{\text{eff}}}{C_{1111}^q} )</td>
<td>( \frac{\beta_{11}^q}{\overline{C}_{1111}^q} )</td>
<td>( C_{1111}^q )</td>
<td>( \beta_1^q )</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>( \frac{C_{2222}^{\text{eff}}}{C_{2222}^q} )</td>
<td>( \frac{\beta_{22}^q}{\overline{C}_{2222}^q} )</td>
<td>( C_{2222}^q )</td>
<td>( \beta_2^q )</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>( \frac{C_{3333}^{\text{eff}}}{C_{3333}^q} )</td>
<td>( \frac{\beta_{33}^q}{\overline{C}_{3333}^q} )</td>
<td>( C_{3333}^q )</td>
<td>( \beta_3^q )</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>( \frac{C_{2323}^{\text{eff}}}{C_{2323}^q} )</td>
<td>( \frac{\beta_{23}^q}{\overline{C}_{2323}^q} )</td>
<td>( C_{2323}^q + C_{2323}^q )</td>
<td>( \beta_2^q + \beta_3^q )</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>( \frac{C_{1313}^{\text{eff}}}{C_{1313}^q} )</td>
<td>( \frac{\beta_{13}^q}{\overline{C}_{1313}^q} )</td>
<td>( C_{1313}^q + C_{1313}^q )</td>
<td>( \beta_1^q + \beta_3^q )</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>( \frac{C_{1212}^{\text{eff}}}{C_{1212}^q} )</td>
<td>( \frac{\beta_{12}^q}{\overline{C}_{1212}^q} )</td>
<td>( C_{1212}^q + C_{1212}^q )</td>
<td>( \beta_1^q + \beta_2^q )</td>
</tr>
</tbody>
</table>
Effective Material Stiffness Tensor Construction of a Plain Weave Lamina

To demonstrate the effectiveness of the MMEs presented, the effective stiffness of a plain weave composite will be formulated using the periodic boundary conditions of an RVE with the loading scenario outlined in Table I. In this section, the geometry is defined and analyzed for each of the geometry independent approaches outlined above, and their results are compared to experimental results from literature and results from a full finite element model meshed with consideration of the weave geometry depicted in Figure 5 and Figure 6.

Geometry

Figure 5: The plain weave geometry as modeled in COMSOL. On the left is the entire system, and in the middle is with the matrix removed. The right pictures shows the quarter-cell geometry to provide clarity on how the strands undulate through the composite. The blue strands are the warp strands and grey the fill strands.

The geometry of the plain weave woven composite used is based on the sinusoidal-based functions presented in Scida, et al. [4]. Figure 5 depicts the geometry as modeled in COMSOL; the full RVE is shown on the left, the resin system is removed for the middle picture, and the right most picture shows a cut view of the strands geometry to help better visualize how the strands undulate. Figure 6 provides a 2D cut plane view of the weave geometry. The geometric functions used in the present study to describe the plain weave composite are defined as

\[
H_f(x_1) = -\frac{h_w}{2} \sin\left(\frac{\pi x_1}{a_w}\right)
\]

(21)

\[
e_f(x_2) = \frac{h_f}{2} \cos\left(\frac{\pi x_2}{a_f}\right)
\]

(22)

The function \(H_f(x_1)\) defines how the fill strand undulates through space and is based upon the fill fiber’s width \(a_f\) and the warp strand’s full thickness \(h_w\). The fill fiber thickness function \(e_f(x_2)\) expresses the thickness of the fill strand as a function of space. In this case, the strand is represented using a sinusoidal cross section (versus a rounded cross section). The functions describing the undulation and thickness of the warp strand \(H_w(x_2)\) and \(e_w(x_1)\), respectively are identical. While more realistic functions exist (see e.g. [10]), a model that could easily and accurately be built and analyzed was desired for the purposes of this study.

Fiber Property Rotation

The strands in the woven composite are transversely isotropic and are variant to rotation. Therefore, the properties of the strand that are defined in the material's principal coordinate axis aligned along the fiber tow need to be rotated into the global coordinate system defined in Figure 7, thus the global direction of the strand undulation is required. The angle of the strand can be found from the undulation function as [4]

---

Page 11
Figure 6: A COMSOL cut-plane representation of the plain-weave woven geometry analyzed with some geometric parameters and functions labeled. This image allows a better view of the strand geometry.

\[
\gamma_f(x_1) = \arctan \left( \frac{dH_f(x_1)}{dx_1} \right) \quad (23)
\]

\[
\gamma_w(x_2) = \arctan \left( \frac{dH_w(x_2)}{dx_2} \right) \quad (24)
\]

Using the angles defined in Figure 7, a stiffness tensor \( C'_{pqrs} \) defined in the local principal direction of the fiber two given by the coordinate angles \((\theta, \phi)\) can be rotated into the global coordinate system via the rotation tensor \( Q(\theta, \phi) \).

\[
C_{ijkl} = Q_{ip}(\theta, \phi)Q_{jq}(\theta, \phi)Q_{kr}(\theta, \phi)Q_{ls}(\theta, \phi)C'_{pqrs} \quad (25)
\]

\[
Q(\theta, \phi) = \begin{bmatrix} 
\sin \theta \cos \phi & \sin \theta \sin \phi & \cos \theta \\
-\sin \phi & \cos \phi & 0 \\
-\cos \theta \cos \phi & -\cos \theta \sin \phi & \sin \theta 
\end{bmatrix} \quad (26)
\]

These rotations are taken into account at every Gauss point being evaluated. When the location of a Gauss point is passed from the element file to the geometry file, the angle of the strand is also returned. This angle represents the angle of the strand with respect to the \(x_1-x_2\) plane, so in order to get \( \theta \) of the rotated system used in \( Q(\theta, \phi) \), the angle \( \gamma_f \) or \( \gamma_w \) is subtracted from \( \frac{\pi}{2} \). The calculation for \( \phi \) is even simpler. The fill strand travels in the \( x_1 \) direction, so the properties are aligned with the with the \( x_1 \) axis, which corresponds to \( \phi = 0 \). The warp strand travels in the \( x_2 \) direction, so \( \phi = \frac{\pi}{2} \).
Figure 7: The angles used to describe the rotation tensor \( Q \). This form of rotating a tensor is invariant to the starting angle; it simply rotates the tensor from any starting rotation to the system described by \( \theta \) and \( \phi \).

Results

The composite analyzed is a plain weave E-Glass and vinylester composite. The material properties of the strand and matrix are given in Table III. The properties listed in Table III are for the properties of the fiber tow as defined in the principal coordinate axis of the material. As noted in Equation (25), these will be rotated into the local coordinate system at every spatial point to account for the tow undulation. The geometric parameters for the fill and warp strand are the same, with a strand thickness \( h_f = h_w = 0.05 \text{ mm} \), and a strand width \( a_f = a_w = 0.6 \text{ mm} \).

Table III: The material properties of a fiber tow and the corresponding matrix from [5].

<table>
<thead>
<tr>
<th></th>
<th>Strand</th>
<th>Matrix</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E_{11} )</td>
<td>57.5 GPa</td>
<td>3.4 GPa</td>
</tr>
<tr>
<td>( E_{22} = E_{33} )</td>
<td>18.8 GPa</td>
<td>3.4 GPa</td>
</tr>
<tr>
<td>( G_{12} = G_{13} )</td>
<td>7.44 GPa</td>
<td>1.49 GPa</td>
</tr>
<tr>
<td>( G_{23} )</td>
<td>7.26 GPa</td>
<td>1.49 GPa</td>
</tr>
<tr>
<td>( \nu_{12} = \nu_{13} )</td>
<td>0.25</td>
<td>0.35</td>
</tr>
<tr>
<td>( \nu_{23} )</td>
<td>0.29</td>
<td>0.35</td>
</tr>
</tbody>
</table>

Finite element code was developed in-house in the MATLAB programming environment to perform the analyses. Within the function that calculates the stiffness matrix of the hybrid elements, the analytic geometry functions defined in Equations (20)-(21) are called to determine whether a spatial location corresponding to a specific Gauss point is within the resin or the matrix.

The two places in a finite element code with the greatest processing time are the element stiffness matrix calculation and the matrix solver. It was originally hypothesized that an unusually large number of integration points would be required to obtain the resolution with each MME to achieve an accurate solution; this in turn would significantly increase the processing time. For a standard finite element procedure, about three Gauss points are required in each direction for
accuracy of linear elements (or 27 in total). A study was performed using a simple cubic geometry with an angled boundary to determine how many Gauss points are needed for convergence, the full details of which are in [19]. It was determined that with only four integration points in each direction (64 in total), the solution is converged within 1%. While no direct computational comparison was performed, it is deduced the computational costs of the MME are similar to that of a traditional finite element.

Figure 8 shows a sample result from the full FEA solution for the \( \varepsilon_{22} \) case. The cut planes show \( \sigma_{22} \) throughout the fibers. The stress in the fiber tows are not constant partially because of the geometry, but also because of the transversely isotropic material properties that have to be rotated as the fiber undulates through space.

Table IV shows the resulting effective stiffness components for the plain weave laminate. The first set of results are obtained experimentally by [5] for a similar laminate and the second set of results are obtained by a full mesh over the geometry using the commercial finite element solver COMSOL. The full finite element solution is termed the true solution in Table IV as it contains all geometric and material property behavior of the element without approximations. The next four columns contain the results from each of the MMEs outlined in the preceding section. Notice that for each of the engineering properties the predicted results are in reasonable agreement with the true solution, with the multi-material element seeming to perform as good or better in predicting the effective stiffness.

<table>
<thead>
<tr>
<th></th>
<th>Experimental [5]</th>
<th>Full FEA (True Soln.)</th>
<th>ASE</th>
<th>B-MME</th>
<th>TMC-MME</th>
<th>STC-MME</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E_{11} ) (GPa)</td>
<td>24.8</td>
<td>24.47</td>
<td>24.92</td>
<td>24.92</td>
<td>24.36</td>
<td>24.34</td>
</tr>
<tr>
<td>( E_{22} ) (GPa)</td>
<td>24.8</td>
<td>24.47</td>
<td>24.92</td>
<td>24.92</td>
<td>24.36</td>
<td>24.34</td>
</tr>
<tr>
<td>( E_{33} ) (GPa)</td>
<td>8.5</td>
<td>9.90</td>
<td>10.99</td>
<td>10.85</td>
<td>10.24</td>
<td>10.20</td>
</tr>
<tr>
<td>( G_{12} ) (GPa)</td>
<td>6.5</td>
<td>4.81</td>
<td>4.90</td>
<td>4.90</td>
<td>4.82</td>
<td>4.78</td>
</tr>
<tr>
<td>( G_{13} ) (GPa)</td>
<td>4.2</td>
<td>3.22</td>
<td>3.66</td>
<td>3.60</td>
<td>3.32</td>
<td>3.24</td>
</tr>
<tr>
<td>( G_{23} ) (GPa)</td>
<td>4.2</td>
<td>3.22</td>
<td>3.66</td>
<td>3.60</td>
<td>3.32</td>
<td>3.24</td>
</tr>
<tr>
<td>( v_{12} )</td>
<td>0.1</td>
<td>0.142</td>
<td>0.143</td>
<td>0.143</td>
<td>0.144</td>
<td>0.144</td>
</tr>
<tr>
<td>( v_{13} )</td>
<td>0.28</td>
<td>0.338</td>
<td>0.324</td>
<td>0.326</td>
<td>0.331</td>
<td>0.333</td>
</tr>
<tr>
<td>( v_{23} )</td>
<td>0.28</td>
<td>0.338</td>
<td>0.324</td>
<td>0.326</td>
<td>0.331</td>
<td>0.333</td>
</tr>
</tbody>
</table>

To quantify the error, we construct results from the various geometry independent analyses of the woven composite and compare them to the true solution using the percent relative error definition as

\[
err = \left| \frac{\text{True} - \text{Approximate}}{\text{True}} \right| \times 100\%
\]  

The results for the relative error are given in Table V, and it is clear from the results that the relative error from the STC-MME is nearly as good if not better that all of the other MMEs. Specifically, the STC-MME reduces the error in the shear, one of the objectives of the new element. Something of note is that a comparison of the results from all material independent elements to the true results show reasonable agreement from all methods applied. This trend is also shown in Figure 9, which plots the relative error in engineering properties as compared to COMSOL against the number of elements used to solve the system. It is noted that all methods
converge relatively quickly to a reasonable error, with the multi-material element consistently converging to a lower error.

Figure 8: $\sigma_{22}$ for the $\varepsilon_{22}$ study performed in COMSOL. A series of five cut planes are used to better present the results.

All of the material independent elements struggled with predicting $E_{33}$, the through-thickness modulus, with the new multi-material element performing the best with a 3% error. This error could be caused by lack of mesh refinement in the $x_3$ direction for any of these models, including the “true” model. Mesh refinement was sacrificed in this direction in favor for the planar directions of the geometry.

Figure 9 shows the error in the engineering terms as compared to COMSOL of each method as the number of elements is increased. Qualitatively speaking, the present study performed the best overall. Caselman’s element performs better for $E_{11}$, and the average stress element performs best for $\nu_{12}$. In some cases, such as for the $G_{23}$ and $G_{13}$ terms in the present study, there’s a “bounce-back” in the error, suggesting the value calculated for these terms passed through the true value. If adding more elements to the study causes the solution calculated to move through the true solution, then this may suggest that these elements are converging to a value other than the “true” calculated value.

Table V: Percent relative error as compared to COMSOL for the E-Glass/Vinylester Plain Weave composite

<table>
<thead>
<tr>
<th>%</th>
<th>ASE</th>
<th>B-MME</th>
<th>TMC-MME</th>
<th>STC-MME</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{11}$ (GPa)</td>
<td>1.84</td>
<td>1.83</td>
<td>0.44</td>
<td>0.54</td>
</tr>
<tr>
<td>$E_{22}$ (GPa)</td>
<td>1.84</td>
<td>1.83</td>
<td>0.44</td>
<td>0.54</td>
</tr>
<tr>
<td>$E_{33}$ (GPa)</td>
<td>10.99</td>
<td>9.60</td>
<td>3.44</td>
<td>3.01</td>
</tr>
<tr>
<td>$G_{12}$ (GPa)</td>
<td>13.64</td>
<td>11.82</td>
<td>3.07</td>
<td>0.69</td>
</tr>
<tr>
<td>$G_{13}$ (GPa)</td>
<td>13.64</td>
<td>11.81</td>
<td>3.07</td>
<td>0.69</td>
</tr>
<tr>
<td>$G_{23}$ (GPa)</td>
<td>1.96</td>
<td>1.88</td>
<td>0.26</td>
<td>0.58</td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>4.21</td>
<td>3.72</td>
<td>2.23</td>
<td>1.59</td>
</tr>
<tr>
<td>$\nu_{13}$</td>
<td>4.21</td>
<td>3.72</td>
<td>2.23</td>
<td>1.59</td>
</tr>
<tr>
<td>$\nu_{23}$</td>
<td>0.31</td>
<td>0.47</td>
<td>0.93</td>
<td>0.82</td>
</tr>
</tbody>
</table>
Figure 9: Error in the engineering properties as compared to the COMSOL study versus the number of elements used in the model. For most properties, the present element provides the most accurate results.

Summary and Future Work

Four elements capable of containing multiple materials were presented and compared in this paper. Overall, while all the elements analyzed performed reasonably well, the new element presented in this study and detailed in [19] performed best in woven composite models. Even though the presented element is still not 100% accurate as would be desired, it has been shown that a significant improvement has been made from the currently available methods.

Ideally, these methods would converge to the true finite element solution as more elements or Gauss points were added. Further work is currently being done in the application to other weave types. This method, with the correct geometric functions, can easily be expanded to analyze satin and twill weave composites. This is potentially where the power of the material independent mesh will be best displayed, as these types of weaves are larger and more complicated, they are much harder to mesh in traditional finite element software packages.

This method of applying material properties at the Gauss points is general enough that it can be used to solve other problems, such as heat transfer or thermal expansion problems. Further, more types of composite models (braided composites, short-fiber and long-fiber filled composites, non-orthogonal weaves, etc.) should also be studied with these models to determining their efficacy in other applications.

Acknowledgements

The authors thank L3 Communications for their gracious financial support for this research.
Bibliography