Ingeniería y Ciencia, ISSN 1794–9165
Volumen 5, número 10, diciembre de 2009, páginas 133–153

Finite Element Modeling of Composite Materials using Kinematic Constraints

Modelo de Elementos Finitos dos Materiais Compostos usando Confinamentos Cinemáticos

Modelado de Materiales Compuestos por Elementos Finitos usando Restricciones Cinemáticas

Merlin Barschke[1], David Uribe[2], Oscar E. Ruiz[3], Jens Jensen[4] and Carlos López[5]

Recepción: 08-nov-2008/Modificación: 24-jun-2009/Aceptación: 07-jul-2009
Se aceptan comentarios y/o discusiones al artículo

Abstract

The purpose of this article is to present simulations of the behavior of composite materials based on kinematic restrictions among the fibers themselves and among fibers and the surrounding resine.

In the literature review the authors have found that the kinematic restrictions have not been fully exploited for modeling composite materials, probably due

---

[1] Undergraduate student of Mechanical Engineering in Aeronautics, mbarschke@stud.hs-bremen.de, University of Applied Sciences, Bremen–Germany.

[2] Undergraduate student of Mechanical Engineering, duribega@eafit.edu.co, CAD/CAM/CAE Lab–EAFIT University, Medellín–Colombia.

[3] PhD in Mechanical Engineering, oruiz@eafit.edu.co, CAD/CAM/CAE Lab–EAFIT University, Medellín–Colombia.

[4] PhD in Engineering, Jens.Jensen@hs-bremen.de, University of Applied Sciences, Bremen–Germany.

[5] Master of Sience in Mechanical Engineering, clopez@eafit.edu, CAD/CAM/CAE Lab–EAFIT University, Medellín–Colombia.

to their high computational expense. The purpose of this article is to show the implementation and results of such a model, by using a Finite Element Analysis of geometric restrictions prescribed to the resine and fiber nodes.

Closed analytic descriptions on behavior of layered composite materials are very rare. Many approaches to describe layered composite material are based on the theory of functions C_Z^1 and C_Z^0, such as the Classic Layer Theory (CLT). These theories of functions contain strong simplifications of the material, in particular for woven composites. A hybrid approximation to model composite materials with Finite Elements (FEA) was developed by Sidhu and Averill [1] and adapted by Li and Sherwood [2] for composite materials woven with glass polypropylene.

The present article presents a method to obtain values for the properties of the composed materials. Such values are used to simulate the reinforced woven fibers by applying layered elements in the ANSYS software. Our model requires less simplifications as compared with the theories C_Z^1 and C_Z^0.

In the present article, differing from the model Li and Sherwood, the weaving model is geometrically simulated. A Boundary Representation (B-Rep "Hand" model) with genus 1 (with complex geometry) was used to apply geometric restrictions to the layers resine, fiber, etc., showing to be appropriate to simulate complex structures.

In the future, non linear properties of materials are to be considered, as well as to perform the required experimental work.

Key words: Composite Materials, Geometric Constraints, Kinematic Constraints.

Resumo

A finalidade deste artigo é apresentar simulações do comportamento dos materiais compostos baseados nos confinamentos cinemáticos entre as fibras mesmas e entre fibras e a resina circunvizinha.

Na revisão da literatura os autores encontraram que os confinamentos cinemáticos não estiveram explorados inteiramente modelando materiais compostos, provavelmente devido a seu custo computacional elevado. A finalidade deste artigo é mostrar a realização e os resultados de tal modelo, usando uma Análise de Elementos Finitos das limitações geométricas prescritas aos nós da resina e da fibra.

As descrições analíticas do comportamento de materiais compostos por níveis são muito raras. Muitas aproximações para descrever os materiais compostos por níveis são baseadas na teoria das funções C_Z^1 e C_Z^0, tal como a teoria clássica da camada (CLT). Estas teorias das funções contêm fortes simplificações do material, especialmente para materiais compostos por tecidos. Uma aproximação hibrida para modelar materiais compostos com elementos finitos

(FEA) foi desenvolvida por Sidhu e por Averill [1] e adaptada por Li e por Sherwood [2] para os materiais compostos tecidos com polipropileno de vidro.

O artigo atual apresenta um método para obter valores para as propriedades dos materiais compostos. Tais valores são usados para simular as fibras tecidas reforçadas aplicando elementos por níveis no software de ANSYS. Nosso modelo requer menos simplificações em comparação às teorias C_Z^1 e C_Z^0.

No artigo atual, diferindo dos modelos do Li e do Sherwood, o modelo de tecido é simulado geomètricamente. Uma Boundary Representation (modelo B-Rep "Hand") com genus 1 (com geometria complexa) foi usada pra aplicar limitações geométricas à resina dos níveis, à fibra, etc., mostrando ser apropriada para simular estruturas complexas.

No futuro, as propriedades não–lineares dos materiais devem ser consideradas, e o trabalho experimental requerido deve ser feito.

Palavras chaves: materiais compostos, confinamentos geométricos, confinamentos cinemáticos.

Resumen

El propósito de este artículo es presentar simulaciones del comportamiento de materiales compuestos basado en restricciones cinemáticas entre las mismas fibras y entre las fibras y la resina circundante.

En la revisión de literatura, los autores han encontrado que las restricciones cinemáticas no han sido plenamente explotadas para modelar materiales compuestos, probablemente debido a su alto costo computacional. El propósito de este articulo es exponer la implementación y resultados de tal modelo, usando Análisis por Elementos Finitos de restricciones geométricas prescritas a los nodos de la resina y las fibras.

Las descripciones analíticas del comportamiento de materiales compuestos raramente aparecen. Muchas aproximaciones para describir materiales compuestos en capas son basadas en la teoría de funciones C_Z^1 y C_Z^0, tal como la Teoría Clásica de Capas (CLT). Estas teorías de funciones contienen significativas simplificaciones del material, especialmente para compuestos tejidos. Una aproximación hibrida para modelar materiales compuestos con Elementos Finitos (FEA) fue desarrollada por Sidhu y Averill [1] y adaptada por Li y Sherwood [2] para materiales compuestos tejidos con polipropileno de vidrio.

Este artículo presenta un método para obtener valores para las propiedades de los materiales compuestos. Tales valores son usados para simular las fibras reforzadas tejidas aplicando elementos de capas en el software ANSYS. El presente modelo requiere menos simplificaciones que las teorías C_Z^1 y C_Z^0.

En el artículo presente, a diferencia del modelo Li–Sherwood, el tejido es modelado geométricamente. Una Representación por la Frontera (B-Rep del modelo

“Hand”) con genus 1 (con geometría compleja) fue usada para aplicar restricciones geométricas a las capas de resina, fibra, etcétera, mostrando que es apropiada para simular estructuras complejas.

En el futuro, las propiedades no–lineales de los materiales deben ser consideradas, y el trabajo experimental requerido debe ser realizado.

Palabras claves: materiales compuestos, restricciones geométricas , restricciones cinemáticas.

Glossary

A	-	Cross section
DOF	-	Degrees of Freedom
d	-	Diameter
E	-	Young’s modulus for isotropic material
E_x	-	Young’s modulus in x direction
E_y	-	Young’s modulus in y direction
E_z	-	Young’s modulus in z direction
f_{max}	-	Maximal deflection
F_b	-	Force for bending only load case
F_t	-	Force for tension only load case
G	-	Shear modulus
h	-	height
I	-	Second moment of inertia
l	-	length
R_m	-	Tensile strength
S	-	Shear force
w	-	width
Δd	-	Variation of the diameter
Δl	-	Variation of the length
γ	-	Deformation angle in shear experiment
ϵ	-	Strain
ν	-	Poisson’s Ratio
σ	-	Stress

1 Introduction

The importance of fiber reinforced composite materials in the field of engineering is growing rapidly. Especially the aerospace industry, but also other industrial sectors as the automobile and naval industry are highly influenced by these kinds of materials.

A composite material is a material which consists of two or more components. Fiber reinforced composite materials consist of the fiber which acts as reinforcement and a matrix which holds the fiber in place. Due to the composition of two materials with highly different properties the analysis of components made of such materials is a complex task.

Studying the behavior of fiber reinforced composite materials follows two different approaches. The micro-mechanical approach handles the composite material as a combination of various materials and derives the average properties considering the properties of the single materials of a unit-cell. The continuum approach considers the composite material as a homogeneous material with uniform average properties.

The Finite Element Method is a powerful tool to analyze components made of fiber reinforced materials. FEA software mostly uses two dimensional elements with layer capabilities to simulate fiber reinforced composite materials. These elements require average material properties of fiber and matrix material, such as average Young's modulus, shear modulus and Poisson's ratio. To obtain these properties continuum approaches are customarily used. Also experiments on the real material are used to obtain these required properties.

In this paper hybrid simulated models of woven fiber reinforced materials are introduced. One of this models is used to obtain the combined material properties for the layered element by performing various experiments in the FEA software.

2 Literature Review

Analytical methods to describe the behavior of fiber reinforced materials are using FOURIER expansions which can be solved exactly to obtain closed analytical models of the material. Such Analytical Methods are described by Savithri and Varadan [3].

C^1_Z function theories assume that the displacement is varying continuously differentiable across the thickness, the most common C^1_Z function theory is the Classical Laminate Theory [CLT].

Donadon [4] presented a 3-D micromechanical model based on the CLT for predicting the elastic behavior of woven laminates. By assuming displacement continuity at layer interfaces, C^1_Z function theories can be achieved. A detailed overview of the most important publications about analytical methods and C^1_Z/C^0_Z function theories have been provided by Rohwer [5].

A micro model for fabric composite materials to perform structural analysis was developed by Tabiei [6]. In this approach the analyzed woven composite material is simulated with an equivalent homogeneous anisotropic material in FEM software. A subroutine which interfaces with the FEM software contains the heterogeneous woven composite material model.

A hybrid FEA model to simulate textile composite materials in a stamping process was introduced by Sidhu and Averill [1]. Li and Sherwood [2] modified this approach for the simulation of woven commingled glass- polypropylene composite fabrics and compared the results of the simulation with experiments on the real material. The modeled woven commingled materials consist of fabric yarns which are commingled fiberglass and polypropylene. The polypropylene melts upon heating and infuses the fiberglass, so that no resin is needed. The hybrid FEA model used by Li and Sherwood [2] was the basis to develop the plane fiber approach discussed in this paper.

Multi-layered circuit boards containing of copper and woven fiber reinforced composite layers are modeled in FEA software and simulated for bending by Li and Kim [7]. The procedure to simulate the differed layers of the material with different orthotropic layers in FEA software was relevant for the investigation made in this paper.

Cao and Akkerman [8] studied the mechanical behavior of woven composites cloth material using experiments and benchmarking. Although these investigations were performed on woven composite fabrics without resin, the described behavior turned out to be useful to understand the behavior of the fiber reinforced composite material discussed in this paper, especially for loads applied transverse the fiber direction.

The mechanical behavior of woven composites was analyzed by Ryou and Chung [9], focused on nonlinear and rate dependent asymmetric/anisotropic deformation behavior. The studies of Ryou and Chung [9] are based on the CLT. Djordjevic and Sekuli [10] studied the nonlinear elastic behavior of different carbon fibers. These Investigations are to be used to integrate nonlinear behavior in the models introduced in this paper. A plane stress constitutive law to describe the inelastic nonlinear, anisotropic, and asymmetric mechanical behavior of fiber reinforced composites based on the kinematic hardening model was developed by Kim and Lee [11]. This Paper offers knowledge about the non elastic behavior of fiber reinforced composites and could be used to integrate plastic deformation in the model discussed in this paper. Zhou and Huang [12] developed a method to apply three dimensional woven structures on 3D models which could be used to apply hybrid approaches as provided in this paper to model woven fiber reinforced materials directly on the model.

3 Methodology

3.1 The Material

The analyzed material is a woven carbon fiber reinforced composite material with a uniform number of warp and filling threads (fig. 1). An epoxy resin is used as matrix. The material dimensions and properties are shown in (tab. 1). In order to simplify the problem, a linear Young's modulus behavior is chosen for the epoxy resin, although the stress-strain behavior of such a material is non linear in reality.

Table 1: Properties of the Analyzed Material

	Young Module	**Poisson's Ratio**	**Cross Section**	**Tensile Strength**
Fiber	230000 N/mm^2	0.3	0.115 mm^2	3500 N/mm^2
Resin	3000 N/mm^2	0.3	-	80 N/mm^2

3.2 The two Hybrid FEA Models

Two ways of modeling the woven fiber reinforced material are developed using the FEA software ANSYS.

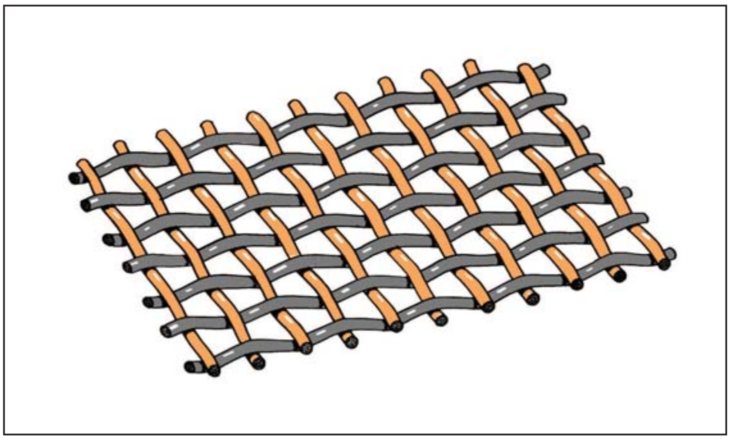

Figure 1: The Woven Structure of the Material

i) The plane fiber approach

ii) The woven fiber approach

3.2.1 The Plane Fiber Approach.

The main objective while developing the first model of the material is to keep the model at a low level of geometric complexity. The plane fiber approach (i) uses ANSYS LINK8 elements to simulate the fibers and SHELL91 elements to simulate the matrix.

In this approach a rigid connections between the orthogonal fibers are established, due to the fact that the vertical and horizontal LINK8 elements use the same nodes. To obtain the correct mass of resin without affecting its moment of inertia, the matrix is modeled in two SHELL63 layers with the fiber in between. In order to connect the two layers of the resin and the fiber layer, each node of the fiber layer is coupled in each DOF with the corresponding nodes of the upper and the lower fiber layer (fig. 2).

3.2.2 The Woven Fiber Approach.

The woven fiber model (ii) is created in order to simulate a more elaborated model which is much closer to reality. Especially the effect of non-rigid connections between the orthogonal fibers, as well as the effect of the curled fiber is to be analyzed here. The woven approach is to be used as a reference model to figure out whether the simplifications made in the plane approach affect the results of stress and strain significantly.

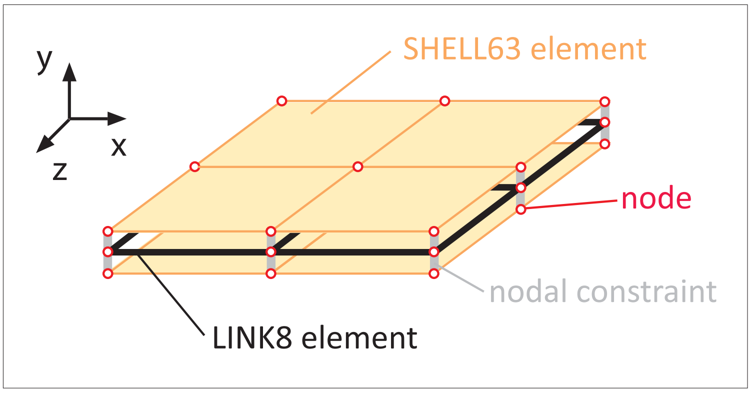

Figure 2: The Plane Fiber Approach

In this approach LINK8 and SOLID45 elements are used to simulate the material, whereas the LINK8 elements represent the fiber and the SOLID45 elements the matrix (fig. 3).

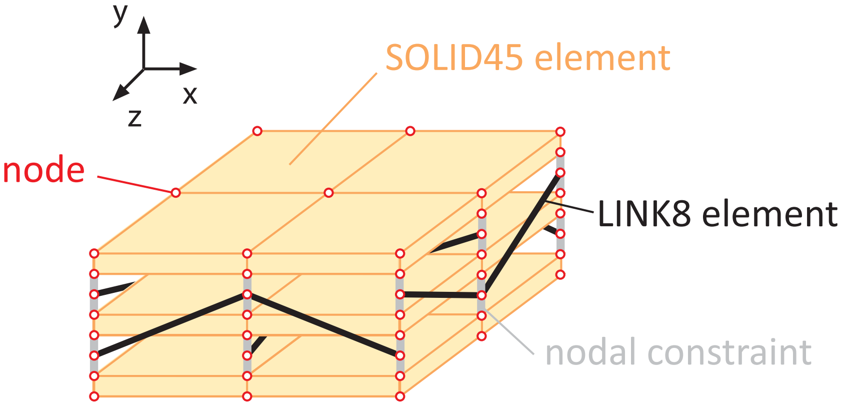

Figure 3: The Woven Fiber Approach

To simulate an adequate role from the resin within the material on one hand, and to keep the model complexity at a reasonable level on the other hand, the resin is divided in three layers. In order to keep the geometrical proportions of the material, the nodes of the LINK8 elements are located centered between the SOLID45 layers. The required rigid connections between the nodes of the LINK8 and the SOLID45 elements, and thus between fiber and matrix, are implemented by coupling the corresponding nodes in all DOF's.

3.2.3 The used ANSYS Elements. The LINK8 element is a one dimensional spar element with tension and compression properties only, which means that no bending stiffness is considered. The element contains six degrees of freedom [DOF]: each of the two nodes of the element is allowed to translate in x, y and z direction. The element is defined by its location of the two nodes, the cross section and the assigned material properties.

The SHELL63 element is a two dimensional, three node (triangular) or four node element (rectangular) with bending and membrane capabilities. Each node has six DOF: translation in x, y and z direction and rotation around the in x, y and z axis. The element is defined by the location of the nodes, the thickness, and the corresponding material properties.

The SOLID45 element is a 3D six node (triangular) or eight node element (rectangular) with three DOF in each node: translation in x, y and z axis. The element is defined by the location of the eight nodes and the corresponding material properties [13].

3.2.4 Comparison of the two Approaches. A MATLAB code which creates a text file in ANSYS Parametric Design Language [APDL] is written for the plane fiber as well as for the woven fiber approach. Reading this text file ANSYS automatically creates all nodes, elements and material properties of the models.

In order to compare the two approaches, one model with an edge length of 29 mm and a thickness of $0,311$ mm is created for each approach. These models are loaded with two differed load cases in order to compare the results of the stresses in fiber and resin and the deformation behavior of the two models. The first load case contains the ANSYS model which is clamped in one edge and loaded with a linear distributed load of 67.67 N/mm on the opposite edge. This load case is chosen to analyze the differences in the stiffness behavior of the straight fiber against the curled fiber.

In the second load case the model is loaded with 100 N on one tip while the two opposite edges are clamped (fig. 4). The results of the comparison are shown in: 4.1 Comparison of the Plane Fiber and the Woven Fiber Approach.

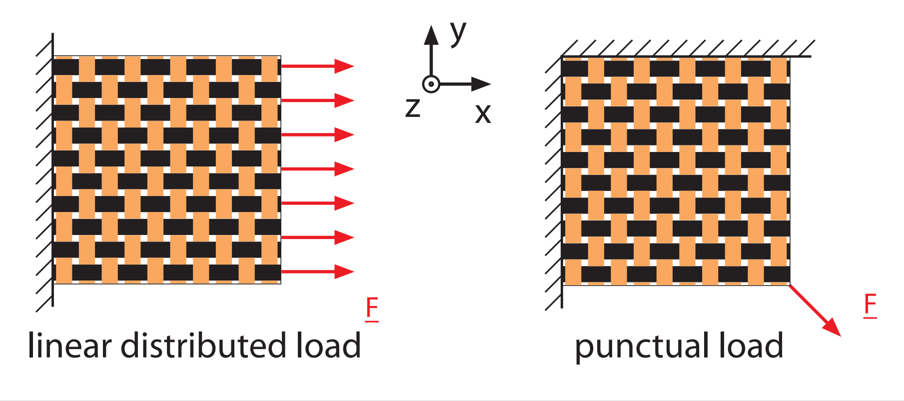

Figure 4: The two Applied Load Cases

3.2.5 Transferring the Material Properties to Shell elements.

The complexity of the woven fiber approach makes it inappropriate to apply it on three dimensional structures. Therefore the material properties of the woven fiber model are to be transferred to a model with a lower complexity, which can be used for three dimensional modeling.

In order to analyze the material properties of the woven fiber approach, two models with different fiber orientations are created, using the programmed MATLAB code (tab. 2 and fig. 5).

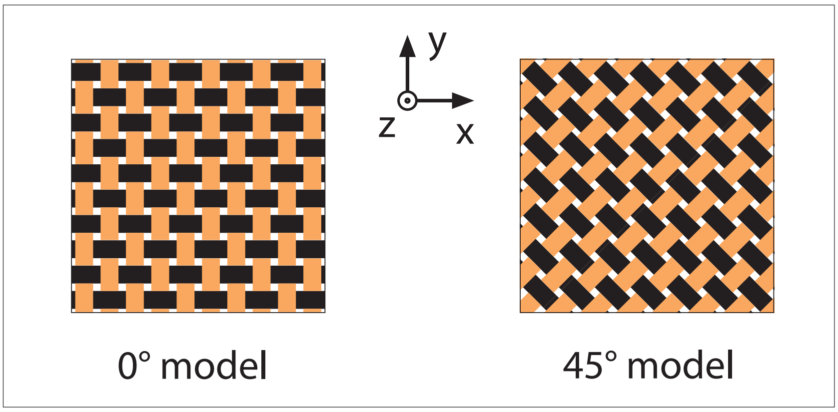

Figure 5: The two Fiber Configurations

A series of load steps provide the linear behavior of force-strain behavior of the models and therefore the linearity of Young's and shear modulus (fig. 6).

Table 2: Properties of the Models with differed fiber configuration

	Edge Length	**Thickness**	**Fiber Orientation**
0°/90° Model	29 mm	0.311 mm	0° and 90°
45°/-45° Model	28.29 mm	0.311 mm	45° and -45°

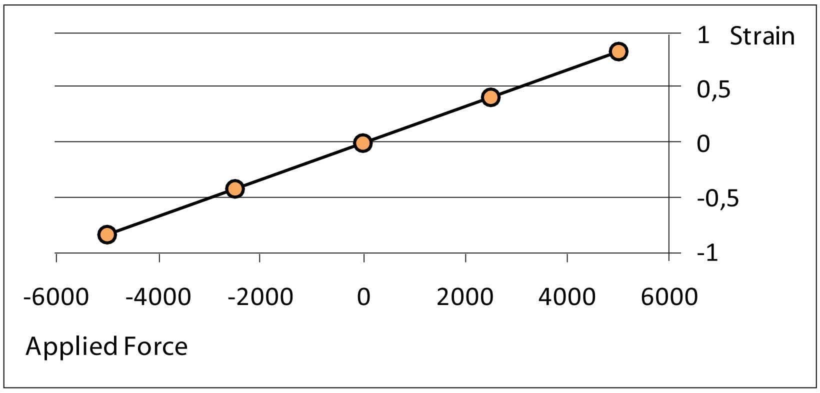

Figure 6: Force Strain Relation

Two load cases are simulated for both models in order to obtain a set of force and displacement each for pure bending and for pure tension. With HOOK's law: $\sigma = E\epsilon$ and equations for the stress in pure tension: $\sigma = \frac{F_t}{A}$. The area is $A = hw$ and the strain is $\epsilon = \frac{\Delta l}{l}$. An equation for the Young's modulus dependent on the Force the area and the displacement valid for pure tension is obtained by (equation 1). The equation for the maximal deflection of a BERNOULLI-beam: $f_{max} = \frac{F_b l^3}{3EI}$, and the equation for the second moment of inertia: $I = \frac{wh^3}{12}$ lead to an equation solving the Young's modulus in the case of pure bending: $E = \frac{12Fl^3}{3f_{max}wh^3}$, which leads to an equation for the height: $h = \sqrt{\frac{4F_b l^3 \Delta l}{f_{max}F_t}}$.

$$E = \frac{F_t}{hw\frac{\Delta l}{l}}\,. \tag{1}$$

With the thickness w, and (equation 1), a number for the Young's modulus for the new material can be calculated. A model with isotropic material properties and this values for Young's modulus and height would have the same characteristic of deformation under pure bending and pure tension as the woven fiber approach model with the 0°/90° fiber configuration. But this is only true for applied forces, that are symmetrical in the xy plane, because the woven model offers anisotropic material properties in the xy plane.

For isotropic materials the shear modulus can be obtained by the equation: $G = \frac{E}{2+2\nu}$. In order to analyze whether the obtained material can be referred to as isotropic, an experiment is performed to obtain the shear modulus and the Poisson's ratio. With the results of a pure tension load case the Poisson's ratio is calculated by the equation: $\nu = \frac{\frac{\Delta d}{d}}{\frac{\Delta l}{l}}$.

With a pure shear model and the equation for the shear: $G = \frac{S}{tan(\gamma)A}$, the shear module is calculated. The obtained values prove the anisotropic behavior of the material.

The same procedure is repeated for the 45°/-45° fiber model, to obtain the same material properties for this model. The 45° angle between the fiber axis and the applied force also causes a displacement component in y direction, besides the displacement component in x direction while a linear distributed load is applied. Since this leads to results which are not valid to calculate the replacement model, additive boundary conditions (no translation in x and z) are established to assure pure deformation in x direction. Such additive boundary conditions are necessary for all of the created load cases, in order to obtain to obtain valid results (fig. 7).

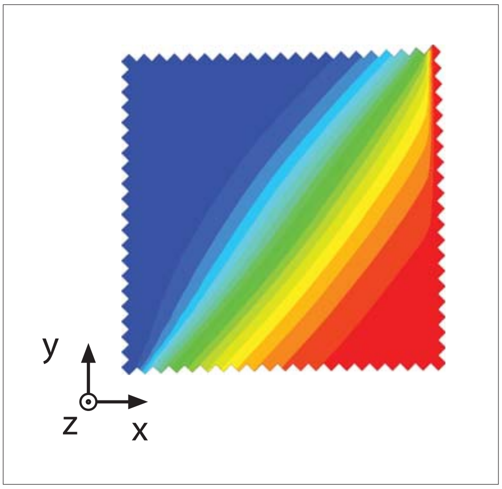

Figure 7: Deformation of the 45°/-45° Model with additional Bounding Conditions

Referred to the obtained properties of the two models as 0° property and 45° property the allocation of these properties in the 0°/90° and the 45°/-45° model can be seen in fig. 8 (tab. 3).

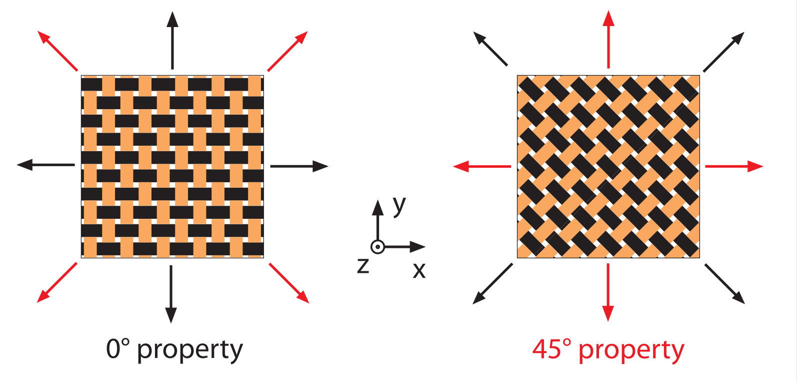

Figure 8: The Material Directional Properties

Table 3: The Calculated Material Properties

	E_x/E_y	E_z	ν	G	h
0° Property	18398 N/mm^2	20078 N/mm^2	0.26	431 N/mm^2	0.48 mm
45° Property	454 N/mm^2	20078 N/mm^2	0.1	4542 N/mm^2	6.94 mm

The obtained values for thickness and Young's modulus are valid for isotropic materials with a given relation between Young's modulus, Poisson's ratio and shear modulus. Since the obtained numbers for these properties are not following these relations, the replacement models must be lightly adjusted manually. For this adjusting three load cases are created (fig. 9). By comparing the results of deformation of the two layer models with the same numbers of the woven approach the Young's modulus E_x/E_y and the shear modulus G are adjusted.

3.3 The Approach for 3D Structures

The three dimensional object which is to be simulated in fiber reinforced composite material is a one genus model of a hand. According to the fact that the hand model contains a number of narrow angles, working with several

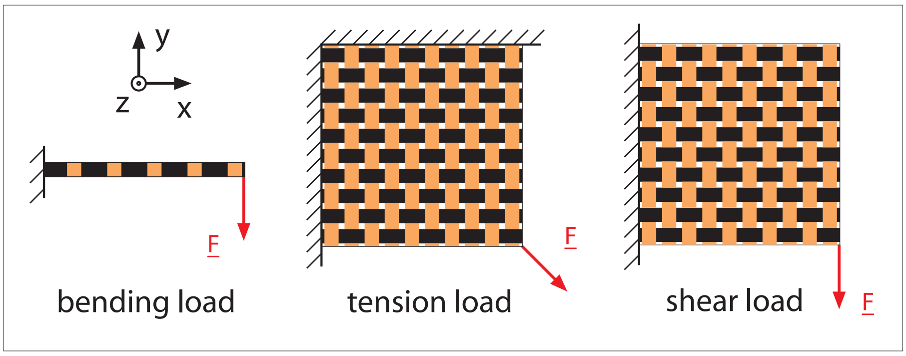

Figure 9: The Applied Load Cases

offset layers would cause many difficulties. Therefore the required material approach should use one layer of nodes only. The shape of the hand is provided as datasets of points (the geometry) and connectivity between the points (the topology).

The provided datasets of the hand lead to triangular elements in ANSYS with randomly orientated elemental coordinate systems. Therefore the elemental coordinate systems would have to be straightened in a uniform direction if one would want to work with directional material properties in the elemental xy plane.

To avoid this problem, the following approach works with isotropic material properties in the elemental xy plane.

For fiber reinforced materials it can be assumed that a model with a 0°, 45°, 90°, 135° fiber configuration has nearly isotropic material-properties in the xy plane. Therefore the material properties of a woven fiber material containing two 45° entwined layers can be assumed as isotropic in the xz plane. Adding this assumption to the further established simplification of the two replacement materials leads to the idea to create a material with one layer of each replacement material with the 0° property and the 45° property. Considering the former established simplifications the only difference between the two layer woven fiber approach and the two layer replacement approach is the varying location of the different material properties in the woven fiber approach.

While in the replacement approach layer one always has the first material property and layer two the other, this distribution varies in reality dependent on the direction of the stress. This leads to the fact that compressive stress and tensile stress are interchanged in one direction in the two layer replacement model in pure bending. Due to the minor thickness of the material and the uniform material behavior in tension and compression the hence resulting error is expected to be negligible.

The stress in the woven fiber approach and the two layer approach not only differ in the numbers, but also in the kinds of stresses in the two models, due to the woven fibers in the first approach which for example creates stresses in z direction in pure tension. Therefore no relation can be found for all the single kinds of stress such as the stress in one coordinate direction or shear stress. Only the von Mieses stress can be compared in the two approaches. By this comparison a factor of 0.35 is obtained by the former experiments with the three load cases (fig. 9) to convert the stress which occurs in the 3D model to the real von Mieses stress in the material.

Since the experiments made have shown that the resin always is the first of the two materials which reaches its tensile strength, the factor conversion is then only necessary to check failure on the resin.

4 Results

4.1 Comparison of the Plane Fiber and the Woven Fiber Approach

ANSYS experiments were conducted to prove the capacity of the geometric - kinematic constraints to account for the improvement in the strength capacity when using woven vs. plain fiber inside a resin matrix. The first experiment includes a distributed linear load (tab. 4). The second, a point load (tab. 5).

Table 4: Comparison of the Results for the Linear Load

	Max. Displacement	**Max. Stress Fiber**	**Max. von Mieses Stress Matrix**
Plane Fiber	0.074981 mm	7.57265 N/mm^2	594.952 N/mm^2
Woven Fiber	0.237999 mm	35.2977 N/mm^2	619.904 N/mm^2

Table 5: Comparison of the Results for the Punctual Load

	Max. Displacement	**Max. Stress Fiber**	**Max. von Mieses Stress Matrix**
Plane Fiber	0.051282 *mm*	79.912 N/mm^2	570.16 N/mm^2
Woven Fiber	0.157359 *mm*	522.874 N/mm^2	502.98 N/mm^2

In both cases the following was found: (a) the deformation with the woven fiber is higher. (b) The maximal stresses carried by the fiber are higher in the woven case. (c) The stresses carried by the

The Woven fiber seems to allow higher flexibility of the structure while the Fiber takes a larger proportion of stress in woven cases, which is expected.

On the other hand, only in the point load the presence of woven fiber seems to be able to reduce the Maximal Von Mieses Stress. The authors consider that additional work is needed to double - check and / or explain such a behavior.

This results prove that the plane fiber approach (i) is not valid to simulate the fiber reinforced composite material in a reasonable accurate way. Therefore the woven approach (ii) is used to obtain the average material numbers for the approach for three dimensional structures.

4.2 Material Properties for the Layered Elements

The finally obtained properties for the two layers are shown in (tab. 6). For each of the former load cases the difference in the numbers of deformation in comparison to the woven fiber approach is less that two percent.

Table 6: The Adjusted Material Properties

	E_x/E_y	E_z	ν	G	h
0°/90° Model	18700 N/mm^2	20078 N/mm^2	0.26	480 N/mm^2	0.48 *mm*
45°/-45° Model	465 N/mm^2	20078 N/mm^2	0.1	4542 N/mm^2	6.94 *mm*

In order to create a FEA model of the hand, a MATLAB code is written which combines the datasets of the hand and the obtained material properties in a APDL file readable for ANSYS.

In order to simulate the two layers of the two layer approach with one layer of nodes, an element is needed which provides layer possibilities. Since the SHELL91 element, which provides such a function is a six node element, the given triangles are imported as areas which are meshed. The allocation of the layers in the layered surface of the 3D models can be seen in (fig 10).

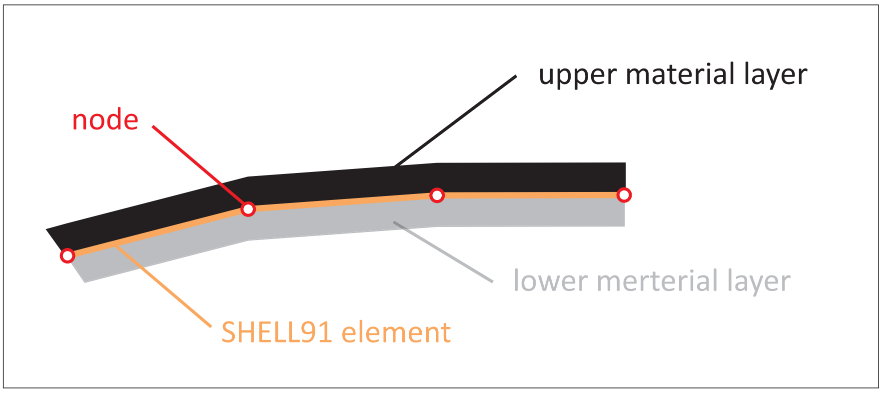

Figure 10: The Layered Surface

Once imported to ANSYS the model of the hand is bounded in each DOF at the bottom and a force is applied on the ring finger of the hand. The hand model, the displacement and the von Mieses stress are shown in (fig. 11).

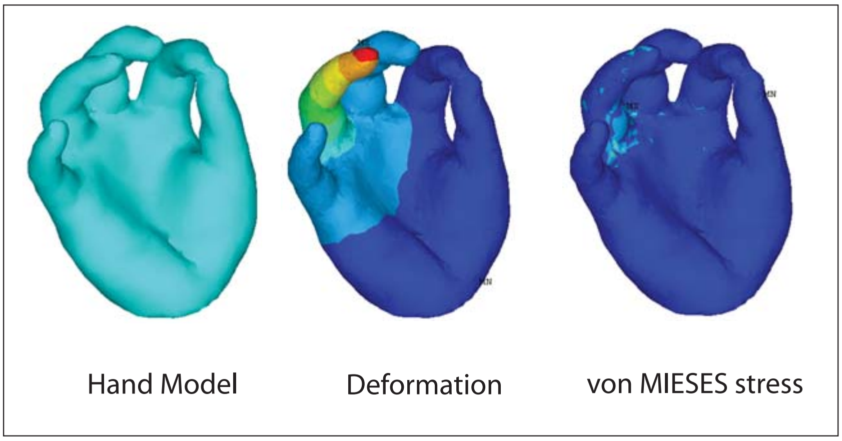

Figure 11: Deformation and Stress in the Hand Model

4.3 Limitations

There are two strong limitations which are to be mentioned.

The first limitation appears due to the chosen linear stress-strain relation of the epoxy resin. In comparison to the linear behavior the nonlinear behavior of the material allows more deformation by the same amount of stress (fig. 11). This leads to the fact that the fiber can borrow a higher amount of the load in reality, so that the nonlinearity of the resins would change the entire material behavior.

The second limitation which has to be considered appears due to the way the average material properties are obtained. Due to the fact that the focus of this procedure was layered on the deformations, the possibilities relate the stresses appearing in the FEA analysis to the real stresses are limited. Only the von Mieses stress can be related to the stress which would appear in reality.

5 Conclusion and Future Work

Two hybrid approaches to model woven fiber reinforced materials were presented in this paper. One of these approaches was used to obtain the material properties for a two layered material which can be applied to a three dimensional structures. To obtain the values of von Mieses stress in the model conversion factors are calculated which relate the von Mieses stress in the modeled material with the real occurring von Mieses stress in the resin.

Since the woven fiber model is based on the assumption of a linear Young's modulus for the resin, nonlinear material properties are to be included in the analysis. The impact of the used simplifications is to be investigated. Also it is to be investigated whether a more elaborated way of obtaining the average material properties can be found, so that all kinds of stresses can be related.

The woven fiber approach is to be compared with real experiments on the discussed material to verify the validity of the approach.

References

[1] R. M. J. S. Sidhu, R. C. Averill, M. Riaz and F. Pourboghrat. *Finite element analysis of textile composite preform stamping*. Composite Structures, ISSN 0263–8223, **52**(3–4), 483–497 (2001). Referenced in 134, 135, 138

[2] Xiang Li, James Sherwood, Lu Liu and Julie Chen. *A material model for woven commingled glass-polypropylene composite fabrics using a hybrid finite element approach*. International Journal of Materials and Product Technology, pISSN 0268–1900, eISSN 1741-5209, **21**(1-2-3), 59–70 (2004). Referenced in 134, 135, 138

[3] T. K. Varadan and S. Savithri. *Laminated plates under uniformly distributed and concentrated loads*. Journal of Applied Mechanics, ISSN 0021–8936, **59**(1), 211–214 (1992). Referenced in 137

[4] Mauricio V. Donadon, Brian G. Falzona, Lorenzo Iannuccia and John M. Hodgkinson. *A 3-d micromechanical model for predicting the elastic behaviour of woven laminates*. Composites Science and Technology, ISSN 0266–3538, **67**(11–12), 2467–2477 (2007). Referenced in 138

[5] K. Rohwer, S. Friedrichs and C. Wehmeyer. *Analyzing Laminated Structures from Fibre-Reinforced Composite Material–An Assessment*. Technische Mechanik, ISSN 0232–3869, **25**(), 59–79 (2005). Referenced in 138

[6] A. Tabiei and Y. Jiang. *Woven fabric composite material model with material nonlinearity for nonlinear finite element simulation*. International Journal of Solids and Structures, ISSN 0020–7683, **36**(18), 2757–2771 (1999). Referenced in 138

[7] L. Li, S. M. Kim, S. H. Song, T. W. Ku, W. J. Song, J. Kim, M. K. Chong, J. W. Park and B. S. Kang. *Finite element modeling and simulation for bending analysis of multi–layer printed circuit boards using woven fiber composite*. Journal of Materials Processing Technology, ISSN 0924–0136, **201**(1–3), 746–750 (2008). Referenced in 138

[8] J. Cao, R. Akkerman, P. Boisse, J. Chen, H.S. Cheng, E. F. de Graaf, J. L. Gorczyca, P. Harrison, G. Hivet, J. Launay, W. Lee, L. Liud, S. V. Lomov, A. Long, E. de Luycker, F. Morestin, J. Padvoiskis, X.Q. Peng, J. Sherwood, Tz. Stoilova, X. M. Tao, I. Verpoest, A. Willems, J. Wiggers, T.X. Yu and B. Zhu. *Characterization of mechanical behavior of woven fabrics: Experimental methods and benchmark results*. Composites. Part A, Applied science and manufacturing, ISSN 1359–835X, **39**(6), 1037–1053 (2008). Referenced in 138

[9] Hansun Ryou, Kwansoo Chung and Woong-Ryeol Yu. *Constitutive modeling of woven composites considering asymmetric/anisotropic, rate dependent, and nonlinear behavior*. Composites. Part A, Applied science and manufacturing, ISSN 1359–835X, **38**(12), 2500–2510 (2007). Referenced in 138, 139

[10] Isidor M. Djordjevic, Daniela R. Sekult and Momcilo M. Stevanovi. *Non-linear elastic behaviour of carbon fibres of different structural and mechanical characteristic*. Journal of the Serbian Chemical Society, ISSN 0352–5139, **72**(5), 513–521 (2007). Referenced in 139

[11] Ji Hoon Kim, Lee Myoung-Gyu, Ryou Hansun, Chung Kwansoo, Jae Ryoun Youn and Tae Jin Kang. *Development of nonlinear constitutive laws for anisotropic and asymmetric fiber reinforced composites*. Polymer Composites, ISSN 0272–8397, **29**(2), 216–228 (2008). Referenced in 139

[12] Kun Zhou, Xin Huang, Xi Wang, Yiying Tong, Mathieu Desbrun, Baining Guo and Heung-Yeung Shum. *Mesh quilting for geometric texture synthesis*. ACM Transactions on Graphics (TOG), ISSN 0730–0301, **25**(3), 690–697 (2006). Referenced in 139

[13] *Documentation for ANSYS*, 11.0 edition, 2007. Referenced in 142