Composites: Part A 94 (2017) 113–123

Contents lists available at ScienceDirect

Composites: Part A

journal homepage: www.elsevier.com/locate/compositesa

A viscoelastic approach for modeling bending behavior in finite element forming simulation of continuously fiber reinforced composites

Dominik Dörr [a,*], Fabian J. Schirmaier [a], Frank Henning [a,b], Luise Kärger [a]

[a] Karlsruhe Insititute of Technology (KIT), Institute of Vehicle System Technology (FAST), Department of Lightweight Technology (LBT), Karlsruhe, Germany
[b] Fraunhofer - Institute of Chemical Technology (ICT), Pfinztal, Germany

ARTICLE INFO

Article history:
Received 30 September 2016
Received in revised form 24 November 2016
Accepted 25 November 2016
Available online 27 November 2016

Keywords:
Forming
Process simulation
Finite element analysis (FEA)
Thermoplastic resin

ABSTRACT

An approach for modeling rate-dependent bending behavior in FE forming simulation for either a unidirectional or a woven/bidirectional reinforcement is presented. The applicability of the bending model to both fiber architectures is guaranteed by introducing either an orthogonal or a non-orthogonal fiber parallel material frame. The applied constitutive laws are based on a Voigt-Kelvin and a generalized Maxwell approach. The bending modeling approaches are parameterized according to the characterization of thermoplastic UD-Tape (PA6-CF), where only the generalized Maxwell approach is capable to describe the material characteristic for all of the considered bending rates. A numerical study using a hemisphere test reveals that the Voigt-Kelvin approach and the generalized Maxwell approach lead to similar results for the prediction of wrinkling behavior. Finally, the approaches for modeling bending behavior are applied to a more complex generic geometry as an application test with a good agreement between forming simulation and experimental tests.

1. Introduction

Forming of two-dimensional pre-products into complexly shaped geometries is one of the most determining process steps in manufacturing of continuously fiber-reinforced composites. Thermoforming of thermoplastic pre-impregnated tapes play an increasingly import role and is currently of great interest especially for the automotive industry due to low cycle times and recycleability [1,2].

During forming, several parameters are influencing the forming process of composites, like temperature, blank holders or grippers, fiber orientation or material behavior. Dependent on these material parameters and process conditions, manufacturing defects like wrinkling, gapping or fiber fracture are feasible [3–6]. Furthermore, a change in fiber orientation and fiber volume content, which has a large impact on structural behavior, is inevitable during forming.

Forming simulation enables an initial validation of the process and the determination of suitable process parameters. This can prevent a time consuming and expensive "trial and error" process design. Furthermore, the fiber reorientation due to forming is predictable, which can be used as input value for structural analyses as well as molding analyses to gain good predictions.

Therefore, constitutive modeling approaches offer the possibility of a detailed analysis of the deformation behavior of engineering textiles or preimpregnated blanks during forming, considering material behavior and process conditions. For this purpose, Finite Element Method (FEM) is highly suitable, as the final shape, as well as the stress and strain distribution can be predicted due to the constitutive modeling of the relevant deformation mechanisms. Namely, relevant deformation mechanisms are the intra-ply mechanisms within a single ply and the interface mechanisms between the plies of the stacked laminate. To model these deformation mechanisms, several codes have been developed in the past ten years. In relation to the current interest on composite forming, some commercial codes are available. Particularly, these are the Pam-Form™ code [7], which is based on an explicit approach, and the AniForm™ code [8], which is based on an implicit approach. Furthermore, several models for composite forming simulation with a clear focus on membrane behavior of a single ply are presented in recent publications, where bending behavior is neglected [9–12]. However, as outlined in numerical studies by several authors [13–15], bending behavior has a distinct influence on the prediction of wrinkles during forming. This is also valid, if blankholders or grippers are employed in the forming process [14], which are used to induce membrane forces during forming.

In general, bending behavior is modeled decoupled from membrane behavior, as bending rigidity is very low compared to fiber

* Corresponding author.
E-mail address: dominik.p.doerr@kit.edu (D. Dörr).

http://dx.doi.org/10.1016/j.compositesa.2016.11.027
1359-835X/

stiffness, and thus conventional plate theory is no longer applicable [8,13–18]. Some models are presented in literature, where bending behavior is modeled in a decoupled fashion purely elastic [8,13,15,17] or elastic and temperature-dependent [16,18]. The AniForm™ code offers the possibility of modeling a Voigt-Kelvin approach for the bending behavior, but no details or application of this modeling approach are available in literature. However, as shown by Sachs [19] and Ropers et al. [20], bending behavior of thermoplastic preimpregnated tapes shows a significant rate-dependency at process conditions.

In this work, a finite strain viscoelastic approach for modeling rate-dependent bending behavior using conventional shell elements is presented in the first part. The presented approach is applicable to uni-directional and bi-directional/woven reinforcements by introducing either an orthogonal or a non-orthogonal fiber parallel material frame. This fiber parallel frame is necessary, as rigid body motion predicted by conventional objective rates lead to a mismatch between the corresponding reference frame and fiber orientation for large shear deformation [9,11,21,22]. The applied material frames are based on the work of Hagège [23], Willems [9] and Badel et al. [21] on the application of hypoelastic constitutive laws for large deformation and strong anisotropy. An alternative elastic approach is presented by Peng and Cao [22] using conventional material frames and a suitable transformation of the stiffness tensor. As shown by several authors [10,11,21], however this approach can lead to artificial strains under shear deformation and is therefore not considered.

The fiber parallel frames are adopted and enhanced in this work for modeling viscoelastic bending behavior decoupled from membrane behavior using an orthotropic Voigt-Kelvin and a generalized Maxwell approach. The approaches are consistent with the work of Zienkiewicz et al. [24], Vidal-Sallé [25] and Rösner [26] on mechanical creep analysis. The bending model is implemented for an implicit and an explicit time integration scheme in the commercially available FE solver Abaqus™. The implicit time integration is applied for the parametrization of the model and the explicit time integration for forming simulation, using the iso-thermal forming simulation framework implemented in several user-subroutines as presented by Dörr et al. [27,28].

In the seccond part of this paper, the bending models are parametrized by means of characterization results of a thermoplastic preimpregnated tape (PA6-CF), which are conducted at the ThermoPlastic composites Research Center (TPRC) in Enschede (NL) by the characterization setup presented by Sachs [19].

In the third part, the parameterized bending models are applied to a numerical hemisphere test, to investigate the influence of the presented modeling approaches on the prediction of wrinkling behavior. Finally, the bending model is applied to the forming simulation of a more complex generic geometry and compared to experimental tests, finished at Fraunhofer ICT (Germany).

2. Finite strain viscoelastic bending model

As mentioned above, a basic requirement on constitutive laws describing the intra-ply mechanisms in FE forming simulation is the decoupling of membrane and bending behavior. Therefore, deformation behavior of a single ply is modeled by applying membrane elements overlaid with conventional shell elements, to facilitate the decoupling of membrane and bending behavior. It has to be taken into account, that shell elements consist of a membrane and a plate part, where only the plate part may be considered in the shell elements. Based on the applied shell theory, a shell element may also contain a transversal shear stiffness, which is not considered in the following, as the presented bending model is based solely on the plate part of the shell element. Considering these aspects, a suitable integration scheme and corresponding constitutive equations are presented in the following.

2.1. Integration scheme decoupled bending behavior

Following shell theory, there is a thickness-based relation between membrane forces $\boldsymbol{N}$ and bending moments $\boldsymbol{M}$, resulting from the integration of the Cauchy stress tensor $\boldsymbol{\sigma}(z)$ at the gauss points of the finite element over the initial thickness t_0 of the shell element by [29]

$$\boldsymbol{N} = \int_{t_0} \{\boldsymbol{\sigma}(z)\bar{f}_{33}\}dz, \tag{1}$$

$$\boldsymbol{M} = \int_{t_0} \{\boldsymbol{\sigma}(z)\bar{f}_{33}^2 z\}dz. \tag{2}$$

The deformation gradient in thickness direction $\bar{f}_{33}$ results from the incompressibility assumption imposed based on the in-plane deformation:

$$\bar{f}_{33} = \frac{1}{\bar{f}_{11}\bar{f}_{22} - \bar{f}_{12}\bar{f}_{21}}, \tag{3}$$

where $\bar{f}_{ij}$ $(i,j = 1,2)$ are the in-plane components of the deformation gradient.

For the description of the strain tensor over the thickness of the shell element, a linear approach is sufficient, as the presented approach is applied to describe bending behavior of a single ply with a high slenderness ratio. Hence, no higher order or discontinuous approach is necessary. Therefore, following Koiter-Sanders shell theory [29], the strain tensor is determined by a linear superposition of membrane strain $\bar{\boldsymbol{\varepsilon}}$ and bending strain $\check{\boldsymbol{\varepsilon}}$, where the bending strain is determined by means of the curvature tensor $\boldsymbol{\kappa}$ of the mid-face and the initial distance z^0 to the mid-face, leading to the strain tensor as a linear function of the initial distance to the mid-face:

$$\boldsymbol{\varepsilon}(z^0) = \bar{\boldsymbol{\varepsilon}} + \underbrace{\bar{f}_{33} z^0 \boldsymbol{\kappa}}_{\check{\boldsymbol{\varepsilon}}}. \tag{4}$$

The Cauchy stress tensor $\boldsymbol{\sigma}$ is determined by means of the constitutive equations depending on the distance to the mid-face. For the decoupling of membrane and bending behavior by superimposed membrane and shell elements, only the bending moments according to Eq. (2) are considered in the shell elements. Therefore, bending behavior is implemented in a (V)UGENS-subroutine, which receives membrane strains and curvature tensor as input and delivers membrane forces (Eqn. (1)) and bending moments (Eqn. (2)) back to the solver. Thus, the decoupling of membrane and bending behavior is facilitated by superimposed built-in membrane elements and a shell elements, where only the bending moments are included.

2.2. Conventional material frames and constitutive equations

Another requirement in FE forming simulation is the consideration of geometric non-linearity due to rigid body motion and large deformation, which mainly results from large shear strains. To account for rigid body motion of the material during forming, it has to be ensured that the stress is described within a material fixed frame, the so-called co-rotational frame, which rotates with the material if it undergoes a rigid body motion. Therefore, an objective rate of the stress tensor $\boldsymbol{\sigma}^{\nabla}$ is introduced, which describes the stress rate of the stress state $\boldsymbol{\sigma}$ within a material fixed frame for an arbitrary rigid body rotation $\mathbf{Q}$ of this frame by

$$\boldsymbol{\sigma}^{\nabla} = \mathbf{Q} \cdot \left(\frac{\mathrm{d}}{\mathrm{d}t}\left(\mathbf{Q}^{\mathrm{T}} \cdot \boldsymbol{\sigma} \cdot \mathbf{Q}\right)\right) \cdot \mathbf{Q}^{\mathrm{T}}. \tag{5}$$