Materials and Design 93 (2016) 431–442

Contents lists available at ScienceDirect

Materials and Design

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

Simulation of thermoplastic prepreg thermoforming based on a visco-hyperelastic model and a thermal homogenization

E. Guzman-Maldonado [a], N. Hamila [a], N. Naouar [a], G. Moulin [b], P. Boisse [a,*]

[a] Université de Lyon, LaMCoS UMR 5259, INSA-Lyon, 27 Av. Jean Capelle, F-69621 Villeurbanne, France
[b] PLASTIC OMNIUM Recherche & développement, Av. du Bois des Vergnes, F-01150 Ste Julie, France

ARTICLE INFO

Article history:
Received 31 October 2015
Received in revised form 28 December 2015
Accepted 29 December 2015
Available online 30 December 2015

Keywords:
Prepreg
Thermoplastic resin
Forming
Thermomechanical
Viscoelasticity
Finite element analysis

ABSTRACT

The simulation of thermoplastic prepreg forming is achieved by alternating thermal and forming analyses. The thermal properties are obtained from a mesoscopic analysis and a homogenization procedure. The forming simulation is based on a viscous-hyperelastic approach. The thermal simulations define the coefficients of the mechanical model that depends on the temperature. The forming simulations modify the boundary conditions and the internal geometry of the thermal analyses. The comparison of the simulation with an experimental thermoforming of a part representative of automotive applications shows the efficiency of the approach.

1. Introduction

Composite materials made of continuous fibers (carbon, glass, aramid, etc.) and polymer matrix are used for their good mechanical properties associated with a low density. The recent commercial and military aircraft make great use of composite materials in their airframes and primary structures. There is also an active interest in this type of materials in the automotive industry to reduce the mass of the components and consequently the fuel consumption of vehicles [1,2]. Nevertheless, the high production rates of these industries require rapid manufacturing processes (typically 1 to 5 min).

Thermoforming thermoplastic prepreg with continuous fiber is a fast process that can be used for car structures manufacturing [3,4]. The process takes a few minutes, and the continuous reinforcements give good material properties. The thermoforming stage of the process example that is analyzed in this paper takes twenty seconds. In addition, composite materials with thermoplastic matrix are more easily recyclable than those using thermoset resins. This topic is important for the automotive industry.

Several parameters affect a thermoforming process of CFRTP (continuous fiber reinforced thermoplastic) composites [5]: temperature, loads on the tools, blank holders, orientation of the plie, etc. Manufacturing double curved parts without defects (wrinkling, porosities, fiber fracture, etc.) can be difficult.

In order to avoid "trial and error" processes, simulation codes for composite manufacturing have been developed to determine and optimize the manufacturing conditions [5–10].

This article presents an approach for the simulation of thermoplastic prepreg thermoforming. The simulation is based on a viscoelastic behavior with a strong thermal dependency. The forming is performed at a temperature slightly higher than the melting point of the resin. Consequently, a small variation of this temperature strongly modifies the mechanical behavior of the prepreg during forming. The simulation is made within a finite element approach. Each ply is modeled as a continuous shell. Continuous models have been proposed for fiber reinforced materials by Spencer [11,12]. These models are based on the fiber inextensibility. They have been extended, in particular, to viscoelastic behavior [13–20].

Given the growing interest in thermoplastic prepreg thermoforming, some commercial codes have been developed for the simulation of these processes. In particular, the Pam-Form™ code [5] based on an explicit F.E.M. approach and the AniForm™ code [18] based on an implicit approach.

Recent papers analyze the thermoforming processes of thermoplastic prepregs. In [21] the influence of the different process parameters was experimentally evaluated. The process simulation presented in [22] is based on a non-orthogonal constitutive model originally developed by Yu et al. [23] and implemented in a user subroutine of code Abaqus Explicit™. The simulations of a double dome geometry thermoforming were compared to the experiments, but wrinkles are not simulated since membrane and truss elements are used. In [18],

* Corresponding author at: INSA Lyon, France.
E-mail address: philippe.boisse@insa-lyon.fr (P. Boisse).

http://dx.doi.org/10.1016/j.matdes.2015.12.166

simulations using AniForm™ software are compared to the experiments in cases of thermostamping unidirectional and woven textile reinforced prepregs. In both works [22,18], the mechanical properties of the prepreg depend on the temperature, but this is assumed to be constant in the blank. However, as it will be shown below, the temperature distribution evolves during forming phase due to the contacts with the tools. The current work aims to present both modeling of the thermal phenomena including the evolution of temperature fields and the simulation of the blank deformation using a finite strain viscoelastic model. The coupling between the thermal and mechanical problems is obtained by performing alternatively both simulations. The thermal properties are obtained from a homogenization procedure taking in count the deformation of the unit cell as a result of the forming. The viscoelastic model is an extension of the work of Simo [24] to the case of thermoplastic prepreg shells with a viscous behavior in in-plane shear.

The thermoforming of an industrial part is carried out both by experimental methods and by numerical simulations based on the presented approach. The comparison between experiments and simulations shows good results.

2. Thermoforming of thermoplastic prepregs

2.1. Stages of the process

The thermoplastic composite thermoforming process consists of four principal stages (Fig. 1). First, the thermoplastic plate is heated by infra-red to a temperature slightly higher than the melting point of the matrix. A robotic arm transports the heated prepreg blank to a mold that is heated to a lower temperature than the blank. This temperature is chosen to ensure a good surface quality and a short cycle time. The mold closes and shapes the composite part. A pressure is then applied in order to crystallize and consolidate the composite. This step must ensure that there are few porosities in the composite part.

2.2. Material

The base material considered in the present study is an 8-Harness Satin Glass/PA66 thermoplastic prepreg. The material is presented as a pre-consolidated blank made out of 5 plies with a [0/90] stacking sequence. The main characteristics of the composite are given in Table A.1 in Appendix A. Its melting temperature is between 257 °C and 265 °C, and the solidification temperature is between 217 °C and 225 °C. This temperature depends on the heating and cooling rates (Fig. 2). After the heating step, the temperature blank is at a uniform temperature of 300 °C and decrease by 3 °C per second by convection. The contact of the blank with the tools leads to a thermal shock that renders a non-uniform temperature distribution in the prepreg.

2.3. Manufactured part

The manufactured part which is analyzed in this work is showed in Fig. 3. It is representative of composite parts for automotive applications. The shape is strongly double curved at the end of the part. The forming needs large shear angles and wrinkles can appear. These wrinkles disrupt the consolidation stage. It is necessary to check that they do not reach the useful part of the composite.

2.4. Objective of the process simulation

The thermoforming simulations aim to determine the feasibility of the process and its conditions [21]. The in-plane shear angles, the possible wrinkles and the compaction/consolidation state are the principal quantities that characterize the thermoforming process, and that will be calculated by the simulation. Because the mechanical behavior is strongly temperature-dependent, thermal analysis and forming simulations are performed alternately. The thermal analysis gives updated parameters of the mechanical behavior. The forming simulation updates the geometry and the tool blank contact zone of the analysis.

The mechanical and thermal material properties must be known. In-plane shear properties are analyzed by a bias-extension test at high temperature [25–27]. Bending properties are determined using a cantilever bending test at temperature [28]. The thermal properties depend on the deformation, on the temperature and on the cooling rate. They are difficult to measure. In this paper, they are determined by a mesoscopic analysis (Section 3). A visco-hyperelastic model is presented in Section 4, the coefficients of this law are all thermo-dependent and updated by the thermal analyses.

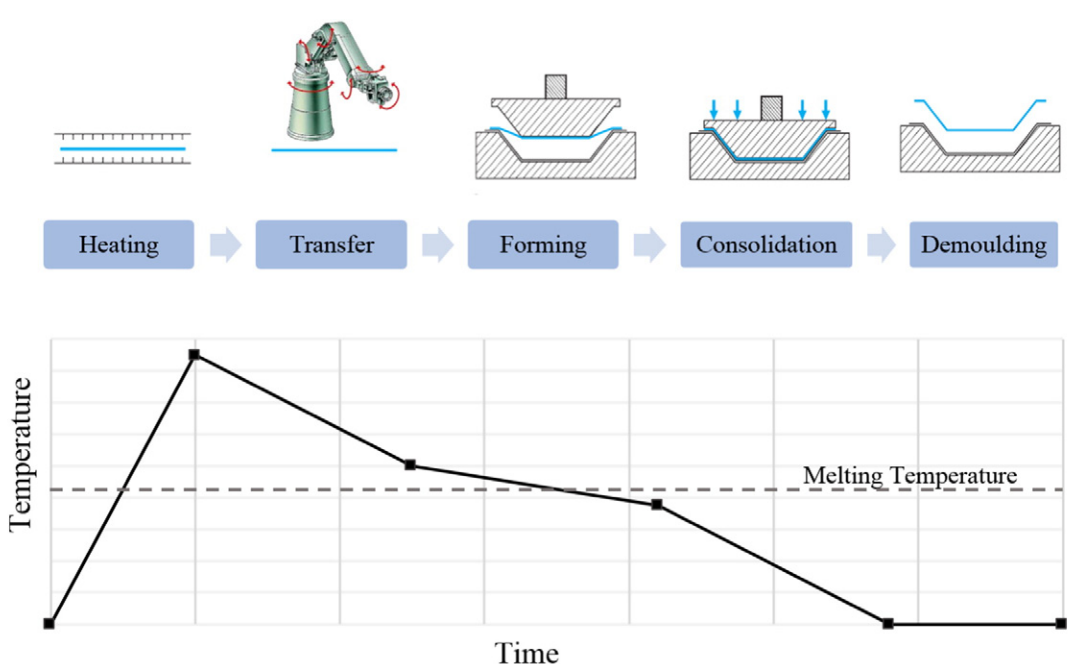

Fig. 1. Stages of the thermoforming process.