Available online at www.sciencedirect.com

ScienceDirect

Procedia Manufacturing 22 (2018) 65–72

Procedia MANUFACTURING

www.elsevier.com/locate/procedia

11th International Conference Interdisciplinarity in Engineering, INTER-ENG 2017, 5-6 October 2017, Tirgu-Mures, Romania

# Reverse engineering in finite element analysis of the behaviour of lignocellulosic materials subjected to cyclic stresses

Mariana Domnica Stanciu*[,a], Amalia Florentina Ardeleanu[a], Horațiu Teodorescu Drăghicescu[a]

[a]*Transilvania University of Brasov, Dep. of Mechanical Engineering, B-dul Eroilor nr. 29, Brasov, 500036, Romania*

**Abstract**

In all engineering fields, continuous improvement of products and processes is being pursued. Reverse engineering is a fast and innovative method of converting a physical model into a virtual one based on its physical and mechanical characteristics. The paper aims to develop the geometric and numerical virtual model of the behavior of a lignocellulosic material subjected to cyclical stresses (tensile-compression) based on mechanical characteristics determined through experimental tests. The importance of this study consists of identifying the simulation method by comparing the experimental results with the numerical results (engineering objective) and developing the working methodology with effect in the education for sustainable development. Modeling with the finite element method (FEM) based on the physical model aimed to determine the results spectrum which are interpolating with the experimentally ones. Thus, the CAD model of the real sample was made by attributing its physical and elastic characteristics using the HyperMesh (preprocessing) and HyperWorks software (post-processing). The model was meshed into QUAD4 elements, after which it were simulated the cyclical stresses on tensile-compression in successive stages, keeping in memory of each stage the changes made at the structural level in the previous stage. Finally, the theoretical results were compared with the experimental ones. The obtained results provide information about distribution of stresses and strains on each layer of material according to the type of stress, traction or compression, for each loading cycle. The values obtained with FEA are close to the experimental ones. The minor differences are due to using an approximate decrease of the modulus of elasticity after each stress cycle. To obtained the real values of Young's Modulus after each cycle would have been necessary a big consumption of samples. The reverse engineering method has the advantage of material economy and the ability to carry out a large number of numerical models to replicate the material's behavior more accurately to cyclical loading and even improve its properties by designing a new material.


Peer-review under responsibility of the scientific committee of the 11th International Conference Interdisciplinarity in Engineering.

* Corresponding author. Tel.: +40-740-555-566; fax: +40-268-410-525.
*E-mail address:* mariana.stanciu@unitbv.ro

2351-9789 © 2018 The Authors. Published by Elsevier B.V.
Peer-review under responsibility of the scientific committee of the 11th International Conference Interdisciplinarity in Engineering.
10.1016/j.promfg.2018.03.011

*Keywords:* Reverse engineering (RE); cyclic loading; lignocellulosic materials; finite element analyses (FEA).

## 1. Introduction

The method called reverse engineering (denoted RE) is based on the partial or total reproduction of the geometric/physical properties of a real model. The process of capturing the digital image of some components is part of the reverse engineering method [1, 2]. Other researchers believe that this method is based on the production of components without the use of an engineering drawing [3, 4], and [5, 6] considered RE to be the development of a new geometry of a product by scanning an existing one and improving the CAD model after digital capture. In this paper, RE is used to analyze stress and strain states of wood samples subjected to cyclic loading – traction and compression, starting from the real test. The purpose of finite element analysis (denoted FEA) was to evaluate the possibility of using FEA for lignocelluloses materials, thus avoiding the consumption of samples and time for experimental tests. Thus, the analysis proceeded from the real behavior of cyclic pine wood samples, geometrically modeling samples and then simulating the cyclical loading. Finally, the spectrum of FEA results was evaluated, interpolated with experimental results.

**Nomenclature**

| | |
|---|---|
| L | length of sample, in mm |
| b | width of sample, in mm |
| h | thickness of sample, in mm |
| $E_L$ | Longitudinal Elasticity Modulus, in MPa |
| $\delta_{EL}$ | Decrement of $E_L$ |
| $G_{RT}$ | Transversal Elasticity Modulus, in MPa |
| $\delta_{GRT}$ | Decrement of $G_{RT}$ |
| $\nu_{RT}$ | Poisson Coefficient |
| $\rho$ | Density, in g/mm$^3$ |
| F | axial loading, in N |
| $\Delta\ell$ | Displacement, in mm |
| $\varepsilon$ | strain, in [%] |
| $\sigma$ | normal stress, in MPa |

## 2. Finite element method applied on lignocellulosic samples

### *2.1. Preprocessing*

In the real test, samples of radially-cut scots pine wood (*Pinus Sylvestris L.),* having the following dimensions: length L = 150 *mm*, width b = 15 *mm*, thickness h = 5 *mm* were analyzed. These were subjected to cyclic loading to tension and compression parallel with fibers (layers) with a force of 1000 *N*, using the LS100 Lloyd's Instruments test machine (Fig. 1, a and Fig. 2). The effective length of the specimen during the test was 100 *mm*. In the preprocessing stage performed in HyperMesh, the sample was modeled as a stratified material with one-dimensional layers of early wood (4 layers) and late wood (3 layers), having different mechanical characteristics (the wider layers represent the early wood, and the narrow - late wood). The material characteristics for each layer are shown in Table 1 and were taken from the literature [7]. Because of the relatively low of thickness and width of the layers, the material card – Mat 1 from HyperMesh library was used in the finite element analysis, which implies that each layer has homogeneous and isotropic properties, and in its entirety the sample will be a layered composite material. As far as boundary conditions are concerned, a grip in the test machine's rims was simulated. For this, the specimen was