FINITE ELEMENTS ANALYSIS OF PLA 3D-PRINTED ELEMENTS AND SHAPE OPTIMIZATION

Paweł Żur, Alicja Kołodziej, Andrzej Baier
Silesian University of Technology, Department of Mechanical Engineering

ABSTRACT
The paper presents the Finite Element Method (FEM) analysis of 3D-printed parts and further shape optimization. Information obtained from the static bending test allowed to carry out the FEM analysis in Siemens NX software with Nastran module. The first part of the research was a comparative analysis of FEM analysis of a single specimen with real-life data collected for polylactide (PLA) specimens made in Fused Deposition Modelling (FDM) technology. Next step was FEM analysis of a PLA brake lever which is used in Silesian Greenpower electric vehicle. Silesian Greenpower team uses 3D-printing for rapid prototyping and manufacturing of customized parts of almost any shape for their car. Purpose of the Finite Element Analysis was shape optimization of the lever in order to ensure adequate safety and ergonomics of use. The strains during its operation should not exceed the maximum permissible for the material used. As this is a key element of safety, a safety factor of level 3 has been adopted. FEM analysis contributed to weight reduction and shape optimization to withstand applied forces.

Keywords: electric vehicle, Fused Deposition Modelling, polymer, simulation, weight reduction