FEM Modeling of a 3D Carbon Fiber Pylon

B. Chiné[1], I. López[1], J. León[1]

[1]Costa Rica Institute of Technology, Cartago, Costa Rica

Abstract

The evolution of specific three dimensional (3D) fabrication printing technologies has developed the capability to fabricate functional parts in several fields of engineering. This has also enabled the possibility to create optimized complex organic structures to use, for example, in new biomedical applications. In this case the lower limb trans-tibial exoprosthesis pylon is the structure of our interest, but there are many other interesting ones. Together with the experimental works, the computational modeling and simulation represent a powerful tool to investigate the performance of these functional printed parts, for assessing the more suitable solutions, as constituent materials, product designs, etc. However, the evaluation, analysis and optimization of these printed parts turns difficult by computational methods, because we need to know the mechanical properties of the materials, for the purpose of modeling and simulating their specific mechanic behavior. The aim of this study is to evaluate the elastic properties of fiber reinforced 3D printed parts under compression stress and compare them with experimental data, in order to study its properties and allow the mechanical analysis by computational tools. Samples of polymer printed parts were fabricated through continuous fiber fabrication (CFF) by using a Markforged Two 3D printer. The principal elastic modulus EZ was obtained by developing experimental tests under the ASTM D695 standard. The transversal modulus was obtained from the theory of composites, by managing specific values of the Poisson modulus and the bulk modulus. These assumptions have allowed later a better approximation of the experimental data.

The modeling work in COMSOL Multiphysics® was developed under static conditions, using the Solid Mechanics Module. The CAD geometries of the prototype printed parts were created in Solid Works and then imported in COMSOL Multiphysics®. Two different CAD models were designed: the first one is a laminated specimen, while the second model is a solid orthotropic material. Appropriate boundary conditions were set in order to model the experimental condition. Finally, by developing the simulating work with COMSOL Multiphysics® we were able to compute the Poisson ratio and bulk modulus, by validating the numerical results with experimental compression tests.

Figures used in the abstract

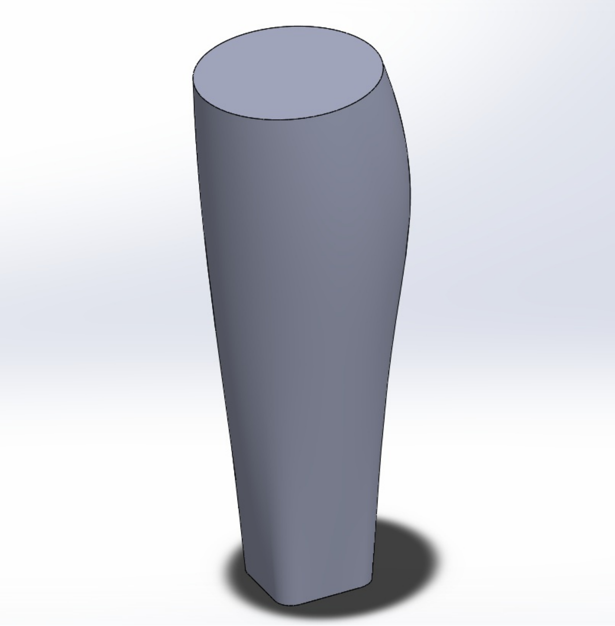

Figure 1: Schematic of the pylon.