

Spring Orthosis Analysis – Finite element modeling and optimization of a composite material

Joakim Berge, Erik Lerneryd and Per Heintz

FS Dynamics Sweden AB

Abstract: This paper covers finite element (FE) analysis and optimization of a spring orthosis, constructed from a pre-impregnated carbon-fibre epoxy composite material. The spring orthosis is one of the most advanced aids that are used in the orthopedist industry. The work has been performed in collaboration with Ortopedteknik, Borås Hospital, at FS Dynamics in Gothenburg.

The purpose of the analyses was to find weaknesses of how the orthosis is built today and to give suggestions of how to change its properties and behaviour. The orthosis has two major interesting areas, the spring and the toe. Both of these areas were analyzed regarding rotation angles of the plies and ply stacking sequence. The geometry of the orthosis, material type and number of composite plies in toe and spring were kept constant during the analysis. The material was modeled as a balanced orthotropic lamina, with a plane stress assumption. To validate the material model a four point bending test of a composite plate was modeled in Abaqus/CAE and compared with experimental data. The interactive Tsai-Wu failure criterion was used as a relative stress measurement. The FE model of the orthosis was created from a scan of the real orthosis. From this the front surface was extracted and used as a thin shell model. The composite layup of the orthosis was created in the way to best imitate the manufacturing procedure used by the engineers at Ortopedteknik.

To be able to analyze the orthosis a walking cycle was recorded from a person wearing the orthosis. This walking cycle was then applied to the model in Abaqus by prescribing reference points and using boundary conditions and multi point constraints. The maximum Tsai-Wu values in the toe and spring part during this walking cycle indicate that the orthosis is not close to failure. The Design Of Experiment (DOE) methodology was used to analyze how the stacking sequences of both the spring (18 plies) and the toe (5 plies) affect the bending stiffness. This was done using the four point bending test in Abaqus where the vertical reaction force was used as stiffness measure. It was found that the angle variation of the outermost plies of the spring has the most effect on the bending stiffness. For the toe the rotation of the two outermost plies are almost alone deciding the bending stiffness of the toe.

From the DOE data approximations were created and then used to optimize stacking sequences based on requested bending stiffnesses.

Keywords: Abaqus CAE, Carbon fibre, Composite material, Composite modeler, Design of experiment, Draping, Finite element analysis, Four point bending test, Isight, Laminate, Optimization, Orthosis, Ply, Shell and Tsai-Wu.