Paper ID: 29945
oral

MATERIALS AND STRUCTURES SYMPOSIUM (C2)
Space Structures I - Development and Verification (Space Vehicles and Components) (1)

Author: Mr. Johannes Boblenz
Deutsches Zentrum fuer Luft- und Raumfahrt (DLR), Germany, Johannes.Boblenz@dlr.de

Mr. Marco Straubel
Deutsches Zentrum für Luft- und Raumfahrt e.V. (DLR), Germany, marco.straubel@dlr.de
Prof. Christian Hühne
German Aerospace Center (DLR), Germany, christian.huehne@dlr.de

SIZING OF THE SANDWICH DOMINATED STRUCTURE OF THE GOSSAMER-1 BOOM AND SAIL DEPLOYMENT UNIT USING ANSYS SOLIDSHELL ELEMENTS

Abstract

During the past two decades, DLR investigated various deployment concepts for solar sails and other huge deployable membrane structures such as drag sails and antennas. One of these concepts is an innovative deployment method for collapsible CFRP-booms and sails that have been developed for the 5 m x 5 m solar sail deployment demonstrator GOSSAMER-1. The deployment is performed by four autarkic deployment mechanisms. These Deployment Units are released from the main module of the sail craft, and deploy each of the flattened and coiled up booms together with the sails in parallel. The development is strongly driven by volume constraints and unusual high launch loads that derive from a generic launch load envelope that was generated to size the payload for a wide bandwidth of potential launchers. A special feature of the BSDU design is that the entire structure is not adhesively bonded but screwed to support the integration process and ensure the late access capability. This paper describes the design of the BSDUs in conjunction with stress and vibration analyses, including modal and PSD analyses. This is done with special regards to the sandwich structure and the screw junctions. Due to the complexity of these connections, the FE-model is based on ANSYS solidshell-elements which provide a better description of these joints in comparison to shell-elements, shown in a submodel comparison between these elements. Furthermore it is shown that the modeling effort of a detailed FE-model is reduced due to the high reusability of the 3D-Geometrie from a CAD-program. Hence, the paper will explain why standard shell elements are not sufficient to describe the complex load distribution in the interface points and will quantify the payoff in more trustworthy simulation results by applying additional joint modelling and calculation efforts.