40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit
11 - 14 July 2004, Fort Lauderdale, Florida

AIAA 2004-3506

Finite Element Analysis of a Composite Overwrapped Pressure Vessel

David L. Gray*
ABAQUS South, Inc., Flower Mound, TX, 75028

Daniel J. Moser†
Compositex, Inc., Sandy, UT, 84092

The application of filament winding technology for the manufacturing of pressure vessels is in widespread use in launch vehicles today. These include solid rocket motor casings as well as liquid fuel and oxidizer tanks for liquid propulsion systems. Typically, composite overwrapped pressure vessels (COPV's) are designed using simplifying assumptions about the load carrying capacity of the fiber filaments in what is commonly referred to as netting analysis. While this method is usually adequate and somewhat conservative for preliminary designs, it does not account for material and geometric nonlinear effects, which are inherent in COPV's. Furthermore, it does not account for the highly orthotropic nature of these structures or give the designer insight into critical areas such as polar boss shear stresses or tangent line stiffness discontinuities and their effects on fiber stresses. A method has been developed to analyze filament wound composite pressure vessels using the ABAQUS finite element analysis program. We compute the orthotropic engineering constants on an element by element basis to accurately predict the structural response of a COPV under internal pressure. In addition, the ABAQUS user subroutine UVARM (user variable) is used to compute the fiber strains during the analysis by means of a coordinate transformation of the strains computed in an axisymmetric model into principle fiber directions. We do this by introducing a user variable (θ), which is the local fiber wind angle. The orthotropic material properties are themselves dependent on the fiber strain due to the tension stiffening effect, which is a characteristic of many graphite fiber materials. By reading the wind angle into user subroutine UVARM, we are able to compute the current fiber strain and use this along with user subroutine USDFLD (user defined field variables) to update the material properties throughout the analysis. This introduces material nonlinearities into the analysis as well as the geometric nonlinearities, which ABAQUS automatically accounts for. The nonlinear analysis is done both with and without the tension stiffened material properties to show their effect on the results.

Nomenclature

$t_{helical}$ = thickness of the helical fibers at the tangent line (no resin)
t_{hoop} = thickness of the hoop fibers on the cylinder (no resin)
P_b = tank design burst pressure
R_{cyl} = radius of the cylindrical section
σ_f = ultimate tensile stress of the fiber
E_i = elastic moduli
G_{ij} = shear moduli

---

* General Manager, 3700 Forums Dr. Suite 101, Member AIAA.
† President and Chief Technical Officer, 11815 Littler Road, Senior Member AIAA.