

Filament Wound Composite Pressure Vessel Analysis with Abaqus

Summary

Filament winding has become a popular construction technique in a wide variety of industries for creating composite structures with high stiffness-to-weight ratios. The difficulty in accurately analyzing the structural behavior of a filament wound body derives from the continually varying orientation of the filaments. The standard capabilities of commercial finite element codes are inadequate to model the spatial variation of fiber orientation in a practical way.

In this Technology Brief, an Abaqus/CAE plug-in for the analysis of filament wound composite pressure vessels is presented. The application allows the user to create, run, and post process a finite element model and allows for detailed specification of structural geometry and winding layout parameters.

Background

In the filament winding composite manufacturing process, filament strands wetted with resin are wound around a rotating mandrel to form a tubular or axisymmetric structure.

Aerospace industry applications of the process include rocket propellant tanks and solid rocket motor casings. Automotive industry applications include high pressure fuel storage tanks for hydrogen powered automobiles.

The construction of filament wound pressure vessels begins with the selection of the underlying liner, which consists of the cylinder and end domes. A polar boss, which is usually constructed of a metal material, is included at the apex of one or both of the domes and is used to mount end plates, rocket nozzles, valves, or other components to the pressure vessel.

Bands of filaments are then wound over the liner in a helical pattern to create layers. After the layers have been wound and cured, the liner is sometimes removed leaving only the filament wound composite structure, while in other cases, the liner remains as part of the pressure vessel. In some designs a rubber shear-ply is placed over the liner near the polar boss region to better accommodate the relatively high shear strains that occur between the composite material and the polar boss.

As the bands of filaments are wound from the cylinder onto the dome, their meridional orientation angle changes from that at the cylinder tangent line to 90 degrees as the



Photo Courtesy of Carleton Technologies Inc, Pressure Technology Division

Key Abaqus Features and Benefits

- Abaqus/CAE customization capabilities, such as custom output processing, and the ability to access and modify model and output data bases through scripting
- Abaqus/CAE modeling capabilities, including automatic surface-based tie constraints between parts, control of mesh quality through partitioning, and geometry import
- Analysis capabilities such as user subroutine `UVARM` to calculate local wind angle and fiber strain output for each element, the inclusion of nonlinear geometric effects, orthotropic and hyperelastic material models, and the ability to model permanent yielding of the liner (autofrettage)

bands wrap around the polar boss and return back up the dome. In order to model this within Abaqus, a unique orientation and material property must be defined for every element within each helical layer. An Abaqus/CAE plug-in application has been developed to automate the entire process of modeling filament wound pressure vessels.

Analysis Approach

The plug-in application allows the user to define all the necessary information to create, run, and post process a finite element model of an axisymmetric wound composite pressure vessel. The application applies a symmetry boundary condition at the mid-length point of the cylinder, so that a half-symmetric model of the complete pressure vessel is created. If a given tank design has different end dome geometries, it can be analyzed by creating two separate models, one for each end of the vessel.

To highlight the features of the application, the analysis of a geodesic-shaped dome pressure vessel, submitted to an internal pressure load, is discussed. Each general step in the process of using the plug-in is explained.

Dome and Cylinder Geometry Definition

Construction of the model begins with the definition of the dome and cylinder geometry. The dome geometry of the pressure vessel can be defined several ways. The plug-in allows the user to define elliptical, spherical or geodesic shapes, or a table of individual points may be entered. Additionally, the geometry can be created from a part instance.

For this brief, the geometry defining the polar boss and liner was created in Abaqus/CAE by importing the necessary geometric parts. The geometry of the cylindrical portion of the structure is defined by entering a cylindrical length within the plug-in dialog box. All other parts of the

model were created by the plug-in. A plot of the final geometry is shown in Figure 1.

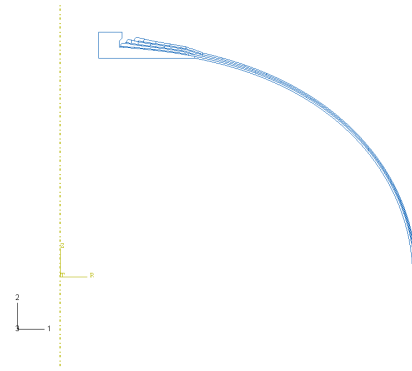


Figure 1: Axisymmetric geometry of wound composite pressure vessel

Winding Layout Specification

After specifying the vessel geometry, the winding layout is defined. The winding layout dialog box is shown in Figure 2. Each layer is assigned a material, wind angle, thickness and band width.

Two additional properties, end type and thickness fraction, control the geometry of the layer at the turnaround point on the dome. The custom application accounts for the thickness buildup as the helical layers approach the turnaround point on the dome.

| Layer Type | Material | Thickness | Wind Angle | Inner Radius | Outer Radius | Band Width | Controls |
|----------------|------------|-----------|------------|--------------|--------------|------------|----------|
| 1 DOILLE | Rubber | 0.03 | | 2.04 | 4.25 | | [Edit] |
| 2 HELICAL_FRIC | T700 | 0.025 | 7.0 | 2.05 | | 2 | [Edit] |
| 3 DOILLE | T700_cloth | 0.03 | | 2.2 | 4.5 | | [Edit] |
| 4 HELICAL_FRIC | T700 | 0.025 | 86 | 12.0 | | 2 | [Edit] |
| 5 HELICAL_FRIC | T700 | 0.025 | 8 | 2.25 | | 2 | [Edit] |
| 6 HELICAL_FRIC | T700 | 0.025 | 87 | 12.06 | | 2 | [Edit] |
| 7 DOILLE | T700_cloth | 0.03 | | 2.65 | 4.75 | | [Edit] |
| 8 HELICAL_FRIC | T700 | 0.025 | 9 | 2.5 | | 2 | [Edit] |

Item is not applicable to current layer type

Figure 2: Winding layout table

A geodesic or non-geodesic winding pattern may be specified using the following equation:

$$\theta(R) = \sin^{-1} \left(\frac{R_0}{R} \right) \pm \delta \left(\frac{R - R_0}{R_{tl} - R_0} \right)^n$$

Here, R is the radial distance from the center line to a point in the layer, R_0 is the radial distance from the centerline to the turnaround point, and R_{tl} is the radius at the dome-cylinder tangent line. A geodesic winding pattern is obtained by choosing $\delta = 0$.

The tank in this example is constructed of T700 graphite/epoxy composite layers using a filament band width of 2.0 inches and a thickness of 0.025 inches. Interspersed between the helical layers are high-angle (hoop) layers which terminate just past the dome-cylinder transition point. Finally, a rubber shear-ply and cloth reinforcement layers are also specified for the region near the polar boss.

The helical layers in a pressure vessel are typically wound in such a manner as to produce an axisymmetric lay-up. In other words, for every helical band oriented at $+\theta$, there is a corresponding band at $-\theta$ to cause the overall laminate to have a balanced angle-ply ($+/-\theta$) lay-up. This assumption is implicit in an axisymmetric model. Therefore only a single orientation angle, defined with respect to the cylinder tangent line needs to be specified for each layer; the plug-in will calculate the angle-ply laminate material properties for each element within each helical layer.

Mesh and Section Creation

The finite element mesh is created automatically by the plug-in. The user may choose the number of mesh seeds through the thickness of the layers and whether to constrain the number of seeds to increase, decrease, or remain constant. A target element aspect ratio may also be entered. The option of choosing a pure quad mesh, quad-dominated mesh, or a triangular mesh is provided. For the present case, a pure quad mesh was chosen. The number of elements through the thickness of the layers was set to 2, but was permitted to be increased by the meshing algorithm. The target aspect ratio was set to 2.0. In addition to these global mesh parameters, individual mesh seeds may be specified to override the global parameters for each individual layer as desired by the user.

When specifying the element section properties, the element order (linear or quadratic) and the element controls (full- or reduced- integration, hybrid or incompatible modes formulation) can be selected. For this case, a linear, reduced-integration element type was chosen. Again, these global settings may be overridden for each layer.

The resulting mesh near the polar boss region is shown in Figure 3. The regions in blue represent the three primary helical layers automatically generated by the plug-in. The region in green represents the polar boss and liner which were imported from an ACIS (.sat) file, then meshed in Abaqus/CAE. The layer of elements in red represents the rubber shear ply doily which was generated by the plug-in, and the cloth doily layers are shown in cyan.

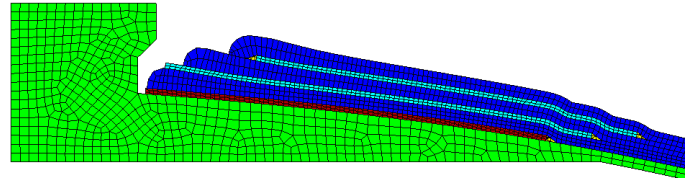


Figure 3: Polar boss region mesh

The mesh near the dome, tangent to the cylinder, is shown in Figure 4. The regions in blue represent the filament wound layers with wind angles of 7.0, 8.0, and 9.0 degrees. The red regions represent the interspersed hoop layers which wrap around the cylindrical portion of the structure. The void regions, where helical layers bridge across ends of hoop layers or doilies, are meshed and assigned resin material properties; these are shown in yellow.

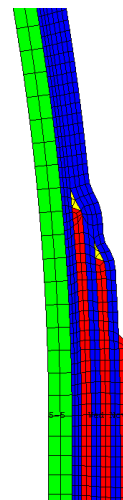


Figure 4: Cylindrical region mesh

Loading

The analysis is carried out in two steps. First, an ultimate pressure load (1.5 x Nominal) of 2250 psi is applied to the liner surface. The aluminum liner experiences yielding in this step. The pressure is returned back to zero in the second analysis step. The residual stresses and strains are captured by the analysis.

Output

While creating the model in the plug-in, the user is given the option of requesting output generated from the UVARM user subroutine. Subroutine UVARM is used for creating user-defined output variables at the material integration points. By default the plug-in automatically creates three output variables; the wind angle (UVARM1), the strain along the fiber direction (UVARM2), and the strain transverse to the fiber direction (UVARM3). By modifying the UVARM subroutine, more user-defined output variables can be created. Because the logarithmic strain is available in the pressure vessel coordinate reference frame and the wind angle is known at each point along the dome, these strains can be transformed into the fiber direction. For this analysis, the default UVARM output generated by the plug-in was used.

Results

Processing the results from the wound composite application begins with the examination of the wind angle throughout the layout to verify that the model was generated properly. A contour plot of the wind angle is shown in Figure 5. The plot shows the expected wind angle layout; specifically, that the angle of each layer sharply approaches ninety degrees as the layer approaches the turnaround point. Additionally, the outermost hoop layer on the cylindrical section clearly shows a wind angle near ninety degrees.

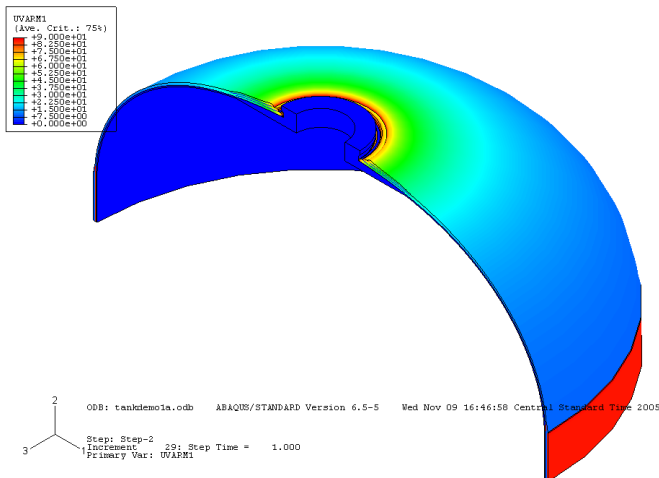


Figure 5: Contour of wind angles (UVARM1)

An alternative method of viewing the wind angle results is to plot the wind angle for each layer as a function of distance along the dome. A special custom output dialog box is included in the plug-in for this purpose. It facilitates the generation of the paths that Abaqus/Viewer requires to generate X-Y path plots. The resulting wind angle plot is shown in Figure 6, again showing the rapid transition at the turnaround radius.

Stresses and strains between the composite layers are of interest because of potential delamination and other inter-laminar failures. The custom path plot utility is again used to plot fiber strain along the interface between layers. Figure 7 displays fiber strain along the innermost nodes of the six layers. The fiber strains are read from user output variable UVARM2.

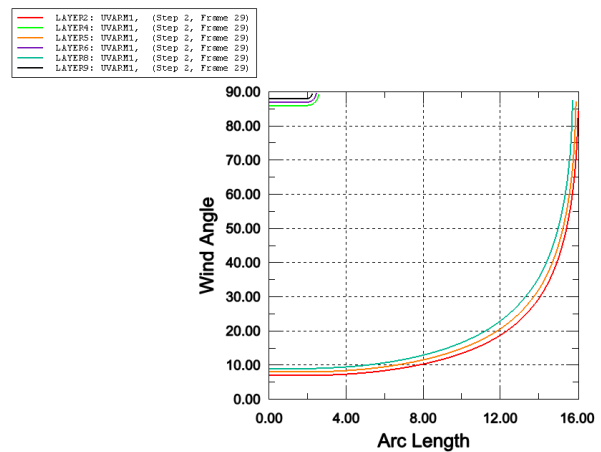


Figure 6: Path plot of wind angles

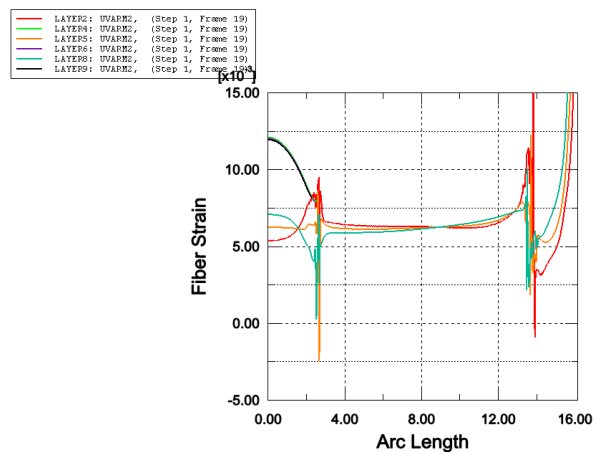


Figure 7: Fiber Strains - Step1 (2250 psi)

Figure 8 shows the fiber strain path plot after unloading (step 2) has occurred. As can be seen, the residual strains are mostly tensile, with some compressive residual strain in the innermost helical layer.

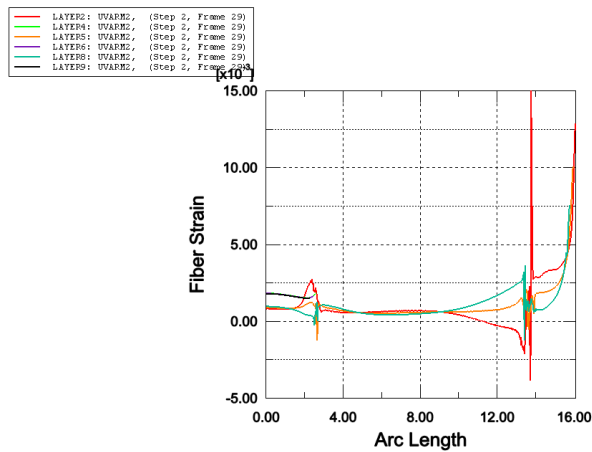


Figure 8: Fiber Strains – Step 2 (0 psi)

Figure 9 shows the fiber strain in the first helical layer at the ultimate pressure load (red) and the residual strain after unloading (green).

Figure 10 shows the equivalent plastic strain (PEEQ) in the aluminum liner after the pressure has been relieved. This permanent plasticity in the liner produces favorable prestress between the liner and the composite overwrap, to help keep the liner from debonding during cryogenic cooling in subsequent loadings as in the case of liquid oxygen storage or other cryogenic applications.

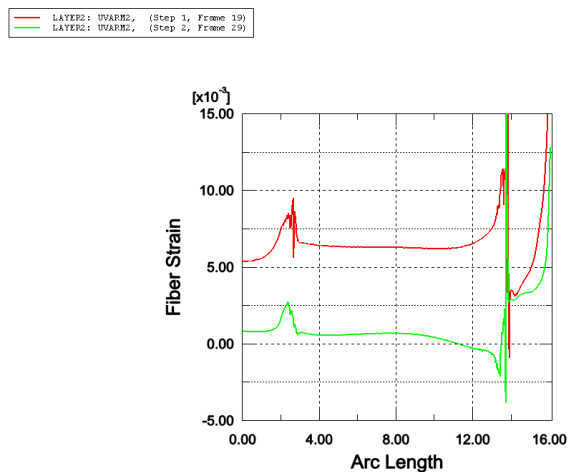


Figure 9: Fiber strain in first helical layer at ultimate pressure and after unloading

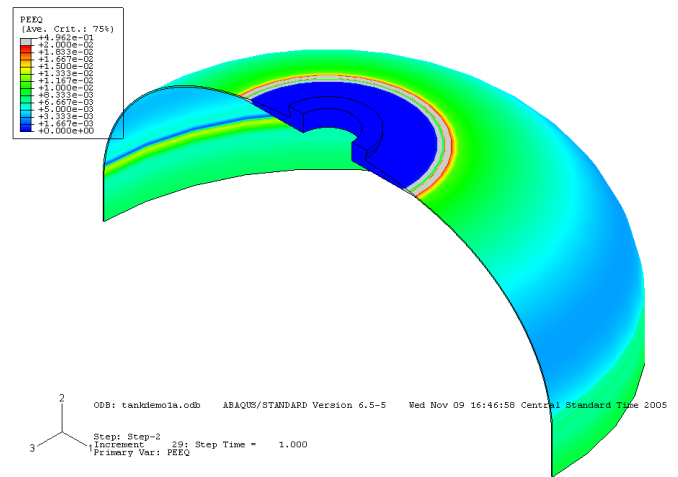


Figure 10: Liner equivalent plastic strain after unloading

Conclusions

The customization capabilities within Abaqus/CAE and the solver capabilities within Abaqus/Standard combine to provide the flexibility needed to create custom applications that can fully automate the analyses of filament wound composite pressure vessels. The full integration between the GUI, the model, and the output database provides a solid foundation for the development of all custom applications.

References

1. Peters, S.T., Humphrey, W.D, and Foral, R.F., Filament Winding Composite Structure Fabrication, 2nd ed.
2. Gray, D.L., and Moser, D.J., "Finite Element Analysis of a Composite Overwrapped Pressure Vessel", American Institute of Aeronautics and Astronautics.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief please see the following Abaqus 6.11 documentation references:

- Abaqus GUI Toolkit User's and Reference Manuals
- CAE User's Manual
 - "The Plug-in toolset," Section 81
 - "Importing and exporting geometry data and models," Section 10
- Abaqus User Subroutine Reference Manual
 - "UVARM" Section 1.1.53
- Analysis User's Manual
 - "Linear elastic behavior," Section 21.2.1

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus, and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include ABAQUS, Inc. Other company, product, and service names may be trademarks or service marks of others.

Copyright © 2007 Dassault Systèmes

