

NASA/TM—1999-209434/REV1



Quantification of Processing Effects on Filament Wound Pressure Vessels

Robert A. Aiello
Modern Technology Corporation, Middleburg Heights, Ohio

Christos C. Chamis
Glenn Research Center, Cleveland, Ohio

National Aeronautics and
Space Administration

Glenn Research Center

March 2002

Document Change History

This printing, numbered as NASA/TM—1999-209434/REV1, March 2002, replaces the previous version in its entirety, NASA/TM—1999-209434, December 1999. It contains the following changes:

Typos were corrected. There were no changes to data.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at <http://gltrs.grc.nasa.gov/GLTRS>

Table of Contents

Summary	1
Introduction	1
Composite Analysis	2
Coordinate Systems	2
Mesh Generation	2
Tape Pretension	3
Linear Generation	3
Polar Wrap Generation	3
Hoop Wrap Generation	3
Helical Wrap Generation	4
Property Input	4
Conclusions	4
Appendix A—Input Deck Setup for Polar-Wound Pressure Vessels	5
Appendix B—Pressure Built-up Due to Pretension in a Polar-Wound Pressure Vessels	7
References	8
Figures	9

Quantification of Processing Effects on Filament Wound Pressure Vessels

Robert A. Aiello
Modern Technology Corporation
Middleburg Heights, Ohio 44130

Christos C. Chamis
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Summary

A computational simulation procedure is described which is designed specifically for the modeling and analysis of filament wound pressure vessels. Cylindrical vessels with spherical or elliptical end caps can be generated automatically. End caps other than spherical or elliptical may be modeled by varying circular sections along the x-axis according to the end cap shape. The finite element model generated is composed of plate type quadrilateral shell elements on the entire vessel surface. This computational procedure can also be used to generate grid, connectivity and material cards (bulk data) for component parts of a larger model. These bulk data are assigned to a user designated file for finite element structural/stress analysis of composite pressure vessels. The procedure accommodates filament wound pressure vessels of all types of shells-of-revolution. It has provisions to readily evaluate initial stresses due to pretension in the winding filaments and residual stresses due to cure temperature.

Introduction

Filament wound pressure vessels, though simple to fabricate, have relatively complex internal structure especially in the domes. Additional complexities are introduced because of the presence of initial stresses. These stresses result from (1) the pretension in the filament tape (wet or prepreg) and (2) the curing process. Accurate descriptions of the local filament location/direction and initial stresses are required for credible evaluation of vessel structural integrity and durability. Computational methods for composite structural stress analyses are available (refs. 1 and 2). These methods have recently been augmented to simulate the local filament location/direction and respective initial stresses. A specific code for that purpose is identified as COBSTRAN for Composite Blade Structural Analysis (ref. 2). The objective of the present article/report is to provide an overview of the COBSTRAN application to filament wound pressure vessels and present some initial results.

The COBSTRAN computer code was developed and used successfully at NASA Glenn Research Center for the design and analysis of composite turbofan (ref. 3) and turboprop blades (ref. 4) and also for composite wind turbine blades (ref. 5). Its application has been extended to include more general models of three-dimensional solid and shell structures and is currently being expanded to include filament wound tank-type structures and/or pressure vessels.

This code combines composite mechanics and laminate theory with a resident data bank of fiber and polymer matrix properties. COBSTRAN is versatile in that it can be used as a preprocessor or post processor with general purpose structural analysis computer codes or stand-alone with its resident general purpose finite element structural analysis module. As a preprocessor for several general purpose finite element structural analysis programs, (COSMIC NASTRAN/MS/NASTRAN and MHOST (refs. 6 to 8)) COBSTRAN, generates a model with anisotropic integrated laminate material properties. Stress or strain output from the analysis program is provided as input to the COBSTRAN post-processor which uses the composite mechanics and laminate theory routines to calculate individual ply stresses, strains, interply stresses, through-the-thickness stresses and failure margins (ref. 1). As an exception, COBSTRAN may utilize nodal force output from MHOST for post-processing. MHOST is the resident general purpose finite element structured analysis module in COBSTRAN.

COBSTRAN, written in FORTRAN 77, is currently configured and tested for operation on various UNIX-based platforms including CRAY X-MP, Silicon Graphics and Hewlett Packard. The COBSTRAN/MHOST code is now available on the PC for automatic pre- and post-processing. The various features of COBSTRAN for filament wound pressure vessels are described in subsequent sections. An input deck setup sample case for a polar winding pressure vessel is illustrated in appendix A. The corresponding pressure buildup due to pretension in the winding filaments is illustrated in appendix B.

Composite Analysis

A COBSTRAN filament winding processing diagram is depicted in figure 1. COBSTRAN is designed to carry out the many linear analyses required to efficiently design and analyze structural components made of multilayered multidirectional fiber composites. Inputs to the COBSTRAN composite analysis are constituent fiber and matrix material properties, factors reflecting the ply fabrication process, and ply orientation and location geometry. COBSTRAN performs the micromechanics, macromechanics, and laminate mechanics of these fiber composites. See the COBSTRAN Theoretical/Programmers Manual (ref. 9). The orientation angle of the 11-axis of a ply is shown in figure 2. For each ply designation this orientation angle is internally calculated for each wrap that exists at a finite element grid point. This angle (measured in degrees) is defined in a local composite coordinate system in which the 0.0 degree axis is tangent to the tank surface and whose projection along the structural x-axis which is parallel to the axial axis of the pressure vessel. The composite 33-axis is positive inward normal to the surface.

Outputs are the various ply and composite properties, composite structural response (accounting for bending-stretching coupling), and composite stress analysis including the results of combined-stress "strength failure criteria" and Hoffman's failure criteria. The output from COBSTRAN post-processing is designed to make use of information generated in the preprocessor phase and stored in temporary binary files. Therefore, the COBSTRAN pre- and post-processing problem must be run in sequence on the same machine.

Coordinate Systems

COBSTRAN utilizes a right hand rectangular coordinate system for structures (vessel) in which the vessel lies axially along the x-axis (fig. 2). For the filament wound tank modeling the input coordinates are assumed to be on the internal surface of the model which corresponds to the mandrel external surface. These coordinates are automatically transferred to the mid-thickness position in accordance with the thickness of the liner and number of wrap ply generated by COBSTRAN.

Mesh Generation

The mesh generation capability in COBSTRAN is designed to create two-dimensional plate type meshes. All meshes are created generally with two-dimensional NASTRAN QUAD4 elements. Under the FWTANK option only quadrilateral elements are created. Tank geometry is defined with user specified circular or elliptical cross-sections by specifying radii and x-axis location. The density of the mesh is controlled by the user input in which the number of circular sections is specified and also the number of elements about the circumference. COBSTRAN calculates the surface geometry and generates a mesh (finite element model) based on this input.

The coordinates of the mesh are first generated on the radii specified by the user input parameters. These dimensions must be the internal dimensions of the vessel since the final thickness of the wraps is not initially known. Following the ply generation and designation of which plies exist at each node, the position of each node is transformed half of the ply stack thickness and normal to the internal surface of the vessel.

Tape Pretension

Filament winding pretension is used to keep the fibers taut and in place in order to prevent fiber wash and fiber wrinkling. However, as the plies build-up the pressure generated by the vessel wall against the mandrel may be high enough to cause mandrel buckling, or liner damage or fiber damage and substantial inner plies wrinkling after mandrel removal. To prevent any or all of those from occurring the winding tension through the ply thickness build-up is varied. This is usually referred to as filament winding or tape programmed pretension.

The external pressure generated by the tape pretension at any ply including the mandrel surface is calculated in accordance with the paper: Residual Stresses in Filament Wound Laminates and Optimum Programmed Winding Tension by C.Y. Liu and C.C. Chamis (ref. 10). Currently the algorithm programmed in COBSTRAN assumes an infinitely stiff mandrel and the pressure is calculated only for hoop plies. The pressure generated by each ply's pretension is accumulated as an external pressure in the finite element model when the input request includes the pneumatic "PRESSURE LOAD."

Liner Generation

A liner, usually a thin metal layer, is used at the inner surface of the shell to prevent seepage of the pressure vessel contents through the composite wall thickness. In COBSTRAN the liner is designated by the key word "LINER" in the wrap stack-up section of the input deck and is treated as another ply. The ply designation number is provided on this input line. The thickness of the liner is indicated on the "CIRCLE," "ELLIPSE," "DSPHERE" or "DELLIPS" lines of the parameter section of the input deck. The number of ply layers required to fill this specified thickness is determined by the code. A sample case input deck setup is illustrated in appendix A. The corresponding pressure build up is illustrated in appendix B. The initial residual stresses are evaluated as described (ref. 11). Both of these must be added to those due to applied internal pressure.

Polar Wrap Generation

A polar wrap is designated by the key word "POLAR" in the wrap stack-up section of the input deck. The angle of the polar wrap is determined by calculating the offset of wrap over the total length of the vessel. The offset is equal to the sum of the radii of both bosses plus the tape width. The polar wrap angle is equal to the arctangent of the offset divided by total vessel length. The number of polar wraps per cycle is determined by dividing the maximum circumference of the tank by the tape width, adjusted for tape overlap and polar wrap angle. The starting angle for a single polar wrap is defined from zero on the positive y-axis incrementing about the x-axis in the positive direction using the right hand rule. If the number of polar pairs is zero only one full circumferential set of wraps will be made. If the number of polar pairs is one, the wrap will be designed as one wrap starting at the designated starting angle and the second wrap starting 180° from the starting position. This feature provides wraps at alternating angles over the vessel for each polar pair indicated. If the number of polar pairs is greater than one, the number of pairs are divided into 360°. Each pair will be designated as one wrap with the starting angle incremented from the designated starting angle and a second wrap starting 180° from the incremented starting position (fig. 3).

Hoop Wrap Generation

A hoop wrap is designated by the key word "HOOP" in the wrap stack-up section of the input deck. The hoop wrap generated by this designation exists only over the cylindrical portion of the vessel. The HOOP ANGLE is measured from a normal to the cylinder x-axis and the tape center line. If the HOOP ANGLE is given as 0.0, the angle of the wrap will be calculated as the arctangent of the tape width, adjusted for overlap, divided by the tank circumference. The number of wraps for each HOOP call is equal to the tape width divided into the circumference. One hoop wrap consists of one wrap in the positive x-direction over the length of the cylindrical section and a wrap in the negative x-direction over the cylindrical section thus generating alternating ply angles. The starting angle for a single hoop wrap is defined from zero on the positive y-axis incrementing about the x-axis in the positive direction

using the right hand rule. The subsequent wraps are incremented by the ratio of the number of wraps divided into 360°. The tape tension generates an equivalent external pressure on the vessel which will exist in the finite element deck if the PRESSURE LOAD option is called in the COBSTRAN input deck (fig. 1). A mandrel pressure buildup for a specific vessel is illustrated in figure 4.

Helical Wrap Generation

A helical winding, designated by the key word "HELICAL" in the wrap stack-up section of the input deck, is generated for a given circuit. A circuit is defined as one complete wrap starting at the boss going around the opposite boss and returning to the original boss. The final position at the starting boss is determined by the tank shape and helical angle used. This final position is the starting position for the next circuit. The incrementing of each circuit and the tape width is included in the program. The helical wrap generated by this designation exists over the entire vessel. The HELIX angle is measured from the cylinder x-axis and the tape centerline. One circuit consists of one wrap in the positive x-direction over the length of the vessel from boss to boss and a continuous wrap in the negative x-direction from boss to boss thus generating alternating ply angles. The starting angle for each helix call is defined from zero on the positive y-axis incrementing about the x-axis in the positive direction using the right hand rule (fig. 5). COBSTRAN/FWTANK can accommodate filament wound shells with other shapes-of-revolution (ref. 12) (see also fig. 6).

Property Input

Two options are available. Option one uses user-supplied ply properties. This option indicates that the unidirectional layer properties are known and will be user-supplied input. Plies are identified by the ply designation number. Option two uses constituent material properties and composite micromechanics to determine ply properties: The appropriate ply designation, the type of fiber and matrix must be specified. Fiber and matrix types from the COBSTRAN resident DATABANK that are most similar to those of the actual ply should be selected. COBSTRAN will use the ply properties exactly as they are entered, but any other material properties will be calculated from the fiber and matrix properties as given in the DATABANK (ref. 1).

Conclusions

An overview is described of a computational simulation procedure to evaluate filament wound pressure vessels. The procedure uses composite mechanics with finite element structural analysis. Unique features of the procedure include internally calculated filament/location direction, dome buildup thickness and pressure build-up due to pretension winding. Polar, helical, and other filament wound patterns are accommodated as well as all types of shells-of-revolution. The initial stresses due to pretension and cure temperature are evaluated from the pretension pressure buildup and the vessel cure temperature. Sample input deck is illustrated in appendix A and pressure buildup in appendix B for a polar wound pressure vessel for a 40-psi vessel the average pressure buildup is 30 psi.

Appendix A

Input Deck Setup for Polar-Wound Pressure Vessels

The following deck setup demonstrates the application of tape pretension for a 5 in. radius cylindrical tank with no end domes. The wall thickness applies only to the liner, the final thickness is determined following the application of all wraps. The grid locations are then translated to the mid-thickness point of the ply stack-up. The 'TRANS' and 'BOUND' parameters apply only to a request for a 'MHOST' model and are ignored by the 'MSC NASATRAN' model selection in the option deck.

```

TITLE= DEMONSTRATION PROBLEM NO. 22B
TITLE= MODELING OF CYLINDER WITH NO DOMES
TITLE= USING EZREAD OPTION AND 'CIRCLE' FORMATS
TITLE= ALUMINUM LINER 0.02 THICK + HOOP WRAPS
TITLE= FIRST HOOP STARTING AT 0.0 DEGREES Y-AXIS
PREPROCESSOR
$DIAG 16 54
$DIAG 98 $ FILL MDES ARRAY WITH WRAP NUMBER NOT COMPOSITE ID
          $ FOR DEBUGGING ONLY: NOT FOR MATERIAL ANALYSIS
DIAG 59 $ DO NOT PRINT INTERNALLY GENERATED CIRCULAR CONTOUR
PRTOUT $ PRINT ALL ELASTIC PROPERTIES
FWTANK $ FILAMENT WOUND TANK MODELING
MHOST $ FORMAT OF THE MODEL GENERATED
$ICAN $ THE ICAN AND MICRO OPTIONS ARE REQUIRED
$MICRO $ FOR USING THE ICAN CODE FOR PRE- AND POST-PROCESSING
ECHO $ DISPLAY THE INPUT DECK
PROPERTY INPUT $ SPECIFY PROPERTIES FOR INNER LINER
PRESSURE LOAD $ GENERATE PRESSURE LOADING DATA
ENDOPTIONS $-----DELIMITER-----
IMAT 1 $ MATERIAL PROPERTY TYPE
NDES 8 $ NUMBER OF PLY DESCRIPTIONS
NYSPACES 14 $ NUMBER OF CHORDWISE SPACES NOT NECESSARILY EQUAL
NOPLY = 8
KSMF = 2 $ STRAIN MAGNIFICATION OPTION FOR POSTPROCESSING
$
$ NAME SECTION SPACES TO X-STA HELIX RADIUS WALL PRESS
$ NUMBER NEXT SEC ANGLE THICK INT/EXT
$
$ GEODESIC 1 8 0.0 25.0 5.0 0.04 1000.0 0.0
$
$ NAME SECTION SPACES TO X-STA RADIUS WALL PRESS
$ NUMBER NEXT SEC THICK INT/EXT
$
$ CIRCLE 1 1 0.0 5.0 .02 00.0 0.0
$ CIRCLE 2 1 2.0 5.0 .02 00.0 0.0
$ CIRCLE 3 1 4.0 5.0 .02 00.0 0.0
$ CIRCLE 4 1 6.0 5.0 .02 00.0 0.0
$ CIRCLE 5 1 8.0 5.0 .02 00.0 0.0
$ CIRCLE 6 0 10.0 5.0 .02 00.0 0.0
$
$ NAME SECTION SPACES TO X-STA HELIX RADIUS WALL PRESS
$ NUMBER NEXT SEC ANGLE THICK INT/EXT
$
$ GEODESIC 15 8 12.0 25.0 5.0 0.04 1000.0 0.0
$
$ NAME SECT ROT RADIAL
$ NUM AXIS DIREC
$
$ TRANS 1 1 2
$ TRANS 2 1 2
$ TRANS 3 1 2

```

Appendix A—Continued

TRANS	4	1	2					
TRANS	5	1	2					
TRANS	6	1	2					
\$ \$ NAME SECT DOF (MAX 6 WITH A SPACE BETWEEN EACH) \$ NUM \$								
BOUND	1	1						
BOUND	7	1						
ENDPARAMETERS \$-----DELIMITER-----								
\$ \$ NAME WRAP PLY TAPE PERCENT START HOOP TAPE \$ STK-ORDER DESIGNATION WIDTH OVERLAP ANGLE ANGLE TENSION \$								
LINER	1		5					
HOOP	2		8	1.0	0.0	0.0	0.0	45.0
HOOP	3		7	1.2	0.0	45.0	0.0	35.0
HOOP	4		8	1.4	0.0	90.0	0.0	25.0
HOOP	5		1	1.6	0.0	135.0	0.0	15.0
ENDWRAPS \$-----DELIMITER-----								
1	T300	EPOX	.005	0.0	.60	0.0		
2	T300	EPOX	.005	0.0	.60	0.0		
3	T300	EPOX	.005	0.0	.60	0.0		
4	T300	EPOX	.005	0.0	.60	0.0		
5	ALT6	ALT6	.010	0.0	.001	0.0		
6	T300	EPOX	.005	0.0	.60	0.0		
7	T300	EPOX	.005	0.0	.60	0.0		
8	T300	EPOX	.005	0.0	.60	0.0		
2								
3								
139.5E+03	0.6E+00	5.595E-02	2.88E+07	1.26E+06	5.94E+05	0.30E+00		
5	117.0E+03	5.62E+03	21.94E+03	7.2E+03	0.011E-06	12.5E-06		
30.0E+03	0.01E-01	0.100E+00	1.00E+07	1.00E+07	5.94E+05	0.30E+00		
	30.0E+03	30.0E+03	30.0E+03	7.2E+03	0.011E-06	12.5E-06		

Appendix B

Pressure Built-Up Due to Pretension in a Polar-Wound Pressure Vessel

Tape Tension Analysis per Chris Chamis paper

a = inside radius of mandrel
 b = outside radius of mandrel
 c = outside radius of laminate
 B = 0 if mandrel is infinitely stiff

E = Young's Modulus
 θ = Tangential direction
 r = Radial direction
 F = Modulus of orthotropic material

$E_\theta = 1.94E+07$ $E_r = 1.38E+06$ $F = 7.33E+05$

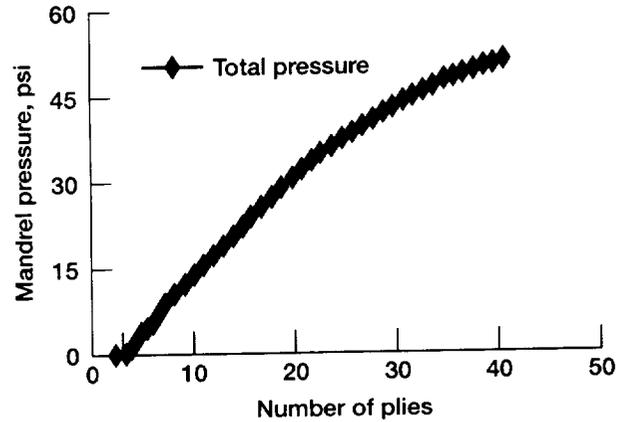
$S = \text{Sqrt}(E_\theta / E_r) = 3.74614$ $\alpha = S - (E_\theta/F) - B = -2.27E+01$
 Mandrel outside rad 5 $\beta = S + (E_\theta/F) + B = 3.02E+01$

Layer Ply thick Radius Tension Pressure Total Pressure

1	0.01	5.01	0	0	0
2	0.01	5.02	0	0	0
3	0.005	5.025	46	2.071602	2.07
4	0.005	5.03	46	2.083424	4.15
5	0.005	5.035	44	2.004071	6.15
6	0.005	5.04	44	2.015228	8.17
7	0.005	5.045	42	1.934205	10.10
8	0.005	5.05	42	1.944714	12.05
9	0.005	5.055	40	1.862052	13.91
10	0.005	5.06	40	1.871931	15.78
11	0.005	5.065	38	1.78766	17.57
12	0.005	5.07	38	1.796925	19.37
13	0.005	5.075	36	1.711072	21.08
14	0.005	5.08	36	1.719739	22.80
15	0.005	5.085	34	1.632333	24.43
16	0.005	5.09	34	1.640417	26.07
17	0.005	5.095	32	1.551484	27.62
18	0.005	5.1	32	1.558999	29.18
19	0.005	5.105	30	1.468565	30.65
20	0.005	5.11	30	1.475525	32.12
21	0.005	5.115	28	1.383615	33.51
22	0.005	5.12	28	1.390034	34.9
23	0.005	5.125	26	1.296672	36.20
24	0.005	5.13	26	1.302562	37.50
25	0.005	5.135	24	1.207771	38.7
26	0.005	5.14	24	1.213146	39.92
27	0.005	5.145	22	1.116949	41.0
28	0.005	5.15	22	1.121819	42.16
29	0.005	5.155	20	1.024238	43.18
30	0.005	5.16	20	1.028616	44.21
31	0.005	5.165	18	0.929672	45.14
32	0.005	5.17	18	0.933568	46.07
33	0.005	5.175	16	0.833282	46.91
34	0.005	5.18	16	0.836707	47.74
35	0.005	5.185	14	0.735099	48.48
36	0.005	5.19	14	0.738063	49.22
37	0.005	5.195	12	0.635153	49.85
38	0.005	5.2	12	0.637666	50.49
39	0.005	5.205	10	0.533472	51.02
40	0.005	5.21	10	0.535544	51.56

$$\text{Press} = (T_0/\beta) \cdot LN \frac{\beta \cdot c \cdot c + \alpha \cdot b \cdot b}{2.0 \cdot b \cdot b}$$

$$B = \frac{E_\theta}{E_m(b^2 \cdot b - a^2 \cdot a)} \left[(1 - \mu_m) \cdot b \cdot b + (1 + \mu_m) \cdot a \cdot a \right]$$



AVERAGE 5.112375 26.6 1.28909 30.03

References

1. P.L.N. Murthy, and C.C. Chamis: "Integrated Composite Analyzer (ICAN)," Users and Programmers Manual. NASA TP-2516, March 1986.
2. R.A. Aiello and C.C. Chamis: "Large Displacements and Stability Analysis of Nonlinear Propeller Structures," NASA TM-82850, May 1982.
3. C.C. Chamis: "Vibration Characteristics of Composite Fan Blades and Comparison with Measured Data." NASA TM X-71893, May 1976.
4. C.C. Chamis, and J.H. Sinclair: "Analysis of High Velocity Impact on Hybrid Composite Fan Blades," NASA TM-79133, April 1979.
5. C.C. Chamis and T.L. Sullivan: "Free Vibrations of the ERDA-NASA 100 Wind Turbine." NASA TM X-71879, March 1976.
6. Anon. NASTRAN User' Manual, NASA SP 222(03), 1978.
7. W.J. Anderson, "MS NASTRAN" Interactive Training Program, John Wiley and Sons, New York, 1983.
8. "3-D Inelastic Analysis Methods for Hot Section Components" (Fourth Annual Status Report), Volume I—Special Finite Element Models, NASA CR-180893, Nakazawa, May 1988.
9. R.A. Aiello and C.C. Chamis: "Composite Blade Structural Analyzer (COBSTRAN)" Theoretical/Programmer's Manual. NASA TM-101958, August 1989.
10. C.Y. Liu, and Chamis, C.C.: "Residual Stresses in Filament Wound Laminates and Optimum Programmed Winding Tension" presented at the 20th Anniversary Technical Conference of the Reinforced Plastic Division of The Society of the Plastics Industry, Inc., pp. 1-10, 1965.
11. C.C. Chamis: "Residual Stresses in Angleplied Laminates and Their Effects on Laminate Fracture." NASA TM-78835, April 1978.
12. R.A. Aiello, "COBSTRAN/FWTAN" Final Report and User's Manual, NASA Contract Report, in publication, 1999.

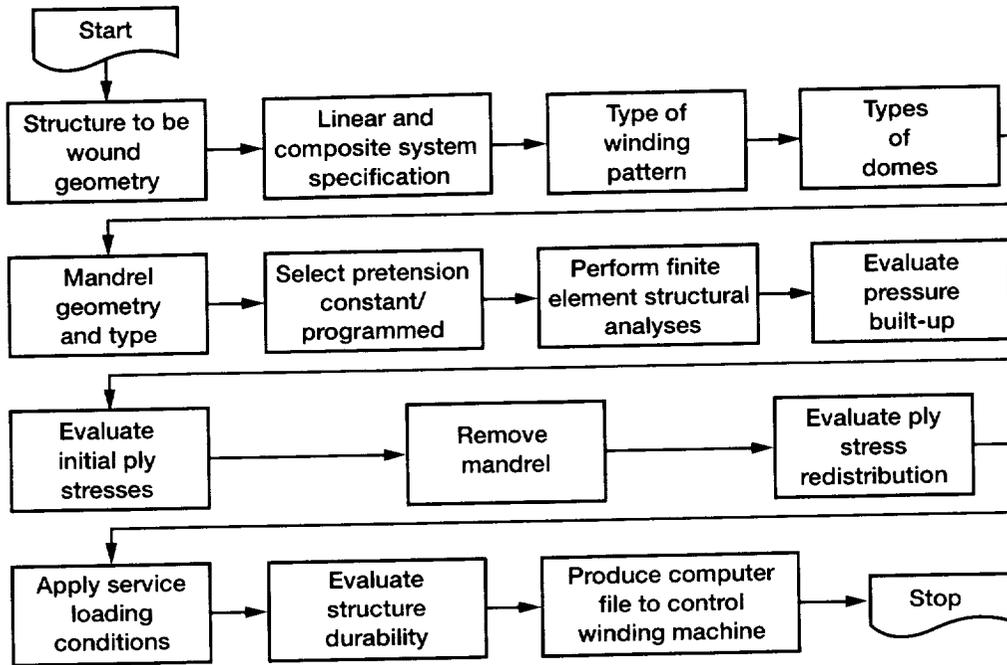


Figure 1.—COBSTRAN filament winding process diagram.

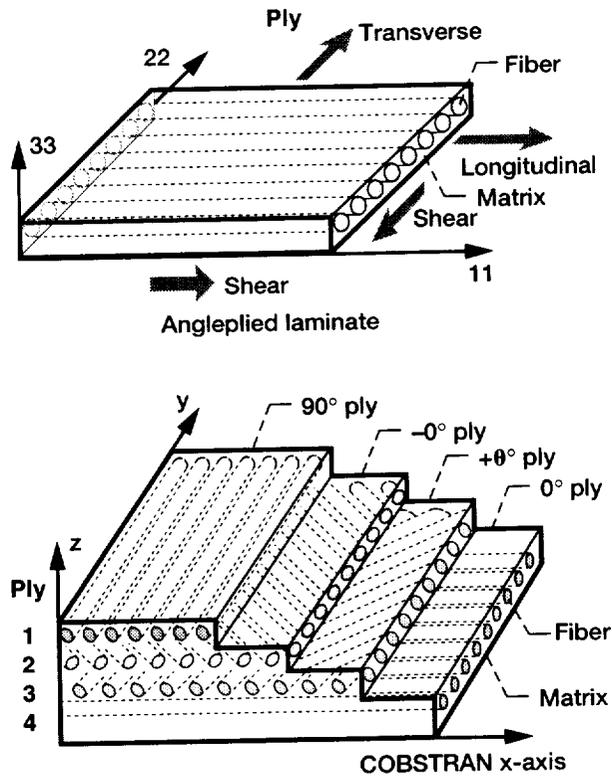


Figure 2.—Fiber composite geometry relative to COBSTRAN coordinate system.

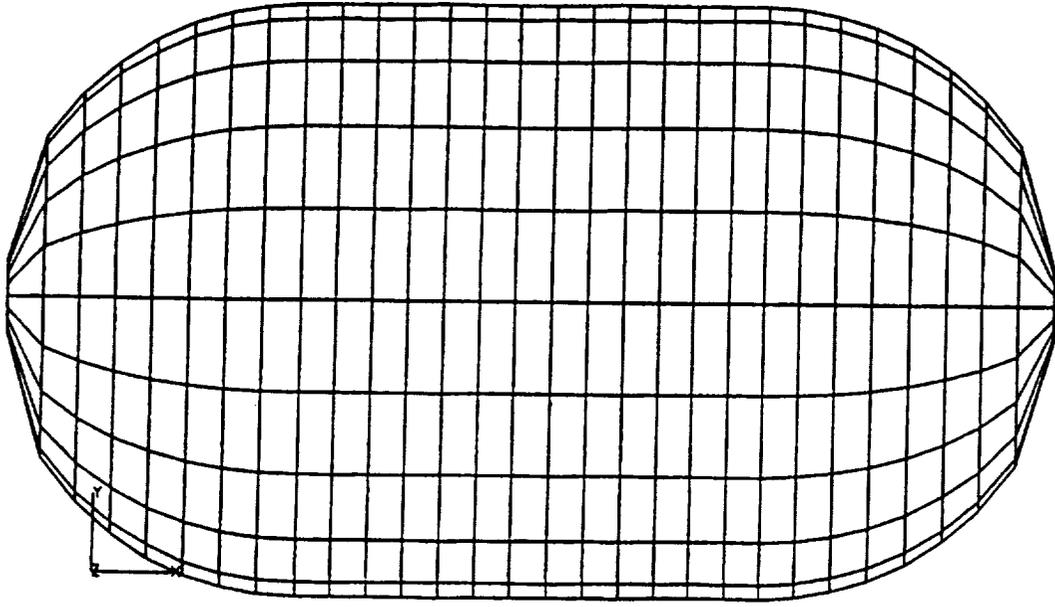


Figure 3.—Tank with spherical endcaps.

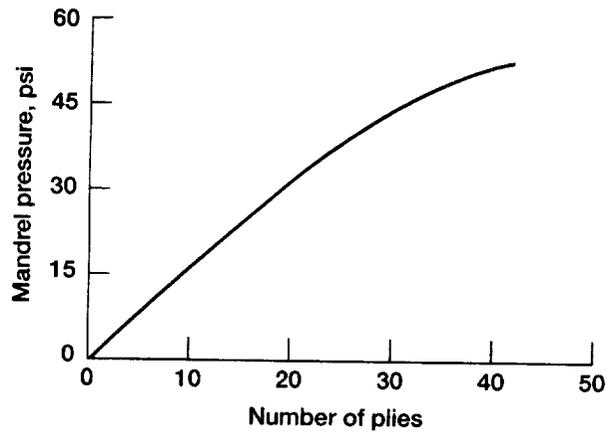


Figure 4.—Mandrel pressure built-up due to filament winding pretension—resulting in initial stresses.

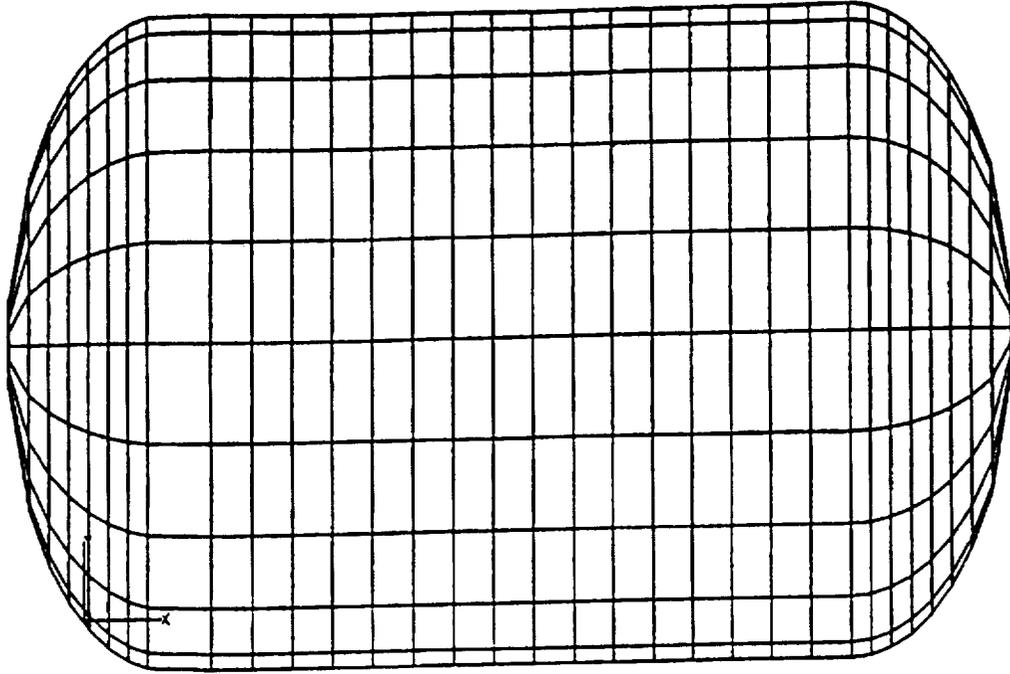


Figure 5.—Tank with elliptical endcaps.

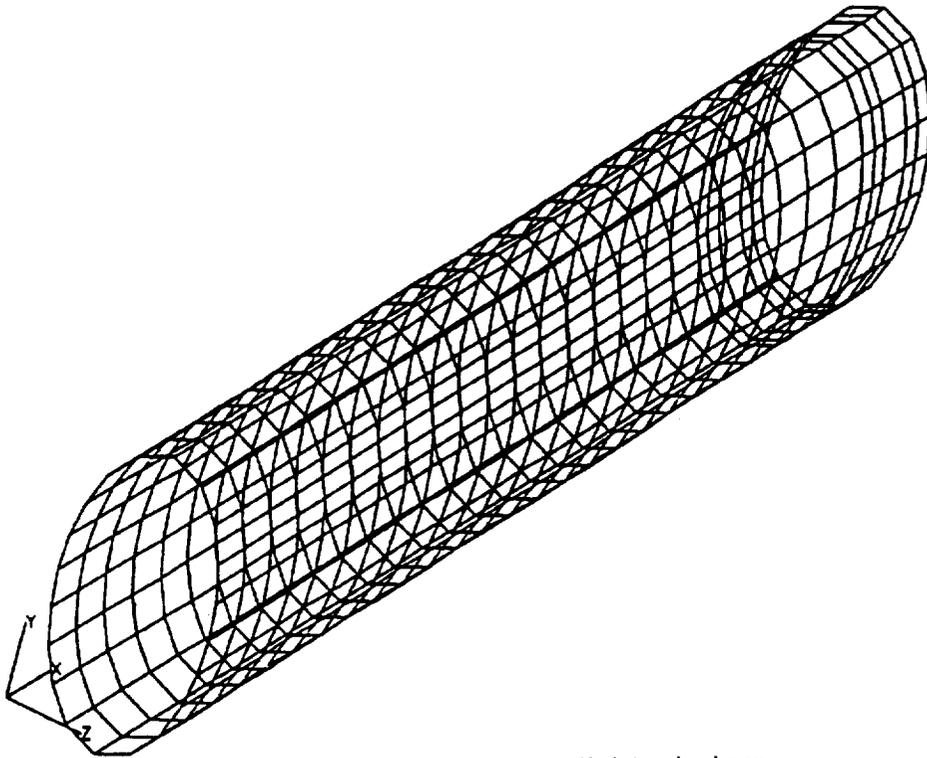


Figure 6.—Elliptical shell used to verify interply shear.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (<i>Leave blank</i>)		2. REPORT DATE March 2002	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE Quantification of Processing Effects on Filament Wound Pressure Vessels			5. FUNDING NUMBERS WU-523-22-13-00	
6. AUTHOR(S) Robert A. Aiello and Christos C. Chamis				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191			8. PERFORMING ORGANIZATION REPORT NUMBER E-11899-1	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-1999-209434-REV 1	
11. SUPPLEMENTARY NOTES Robert A. Aiello, Modern Technology Corporation, Middleburg Heights, Ohio 44130, and Christos C. Chamis, NASA Glenn Research Center. Responsible person, Christos C. Chamis, organization code 5000, 216-433-3252.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category: 39 Available electronically at http://gltrs.grc.nasa.gov/GLTRS This publication is available from the NASA Center for AeroSpace Information, 301-621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (<i>Maximum 200 words</i>) A computational simulation procedure is described which is designed specifically for the modeling and analysis of filament wound pressure vessels. Cylindrical vessels with spherical or elliptical end caps can be generated automatically. End caps other than spherical or elliptical may be modeled by varying circular sections along the x-axis according to the end cap shape. The finite element model generated is composed of plate type quadrilateral shell elements on the entire vessel surface. This computational procedure can also be used to generate grid, connectivity and material cards (bulk data) for component parts of a larger model. These bulk data are assigned to a user designated file for finite element structural/stress analysis of composite pressure vessels. The procedure accommodates filament wound pressure vessels of all types of shells-of-revolution. It has provisions to readily evaluate initial stresses due to pretension in the winding filaments and residual stresses due to cure temperature.				
14. SUBJECT TERMS Composite mechanics; Filament wound; Pretension; Cure stresses; Finite element; Computer codes; Pressure buildup			15. NUMBER OF PAGES 18	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	