II CONGRESSO NACIONAL DE ENGENHARIA MECÂNICA
II NATIONAL CONGRESS OF MECHANICAL ENGINEERING
12 a 16 de Agosto de 2002 - João Pessoa – PB

STRUCTURAL ANALYSIS OF COMPOSITE ROCKET MOTOR CASES

Josef Betten
Department for Mathematical Models in Materials Science (MMW)
Aachen University of Technology (RWTH Aachen), betten@mmw.rwth-aachen.de

Jens Krieger
Department for Mathematical Models in Materials Science (MMW)
Aachen University of Technology (RWTH Aachen), krieger@mmw.rwth-aachen.de

Luís Eduardo Vergueiro Loures da Costa
Centro Técnico Aeroespacial (CTA/IAE/ASE)
Praça Mal.-do-Ar Eduardo Gomes, 50
São José dos Campos (SP), CEP 12228-904, Brazil, loures@iae.cta.br

Jörg Bernhard Multhoff
Department for Mathematical Models in Materials Science (MMW)
Aachen University of Technology (RWTH Aachen)
Augustinerbach 4-22, 52064 Aachen, Germany, multhoff@mmw.rwth-aachen.de

Abstract. *Solid rocket motors can be used as apogee motors for sounding rockets and satellite launchers. The main structure of the motor acts as a pressure vessel and can be manufactured efficiently with composite materials. Design and analysis of these composite cases are considerably more complex than those for motor cases from traditional engineering materials. The finite element method is an adequate tool for the analysis of such complex structures. In the study presented in this paper layered volume elements are used to create a high-resolution model to analyze the stress distribution under internal pressure. Degree of freedom constraints are defined to enforce an axisymmetric solution and to reduce the computational effort. A comprehensive convergence study is performed to determine the necessary mesh density for the required accuracy. The model was applied to verify the behavior of a motor case used in the Brazilian space program and the results are compared with experimental data from hydraulic burst tests.*

Keywords: *Rocket Motor Case, Composite Pressure Vessel, Finite Element Analysis.*

1. INTRODUCTION

Solid rocket motor cases act mainly as pressure vessels and are frequently made from fiber reinforced composites by filament winding. In this way the full potential of composite materials can be exploited. Furthermore the manufacturing process is cost effective. However, design and analysis of composite structures is considerably more complex than the design of structures made from more traditional engineering materials like steel or aluminum. This is because composite structures are usually layered and have anisotropic characteristics, which means that much more information must be specified and analyzed. Furthermore the mechanics of composite structures is complicated by the possible coupling between extensional, bending and shearing deformation modes and the greater variety of failure modes (Jones, 1999). Effective and efficient computational tools are necessary to help in the design of optimized composite structures. The Institute for Aeronautics and Space (IAE) develops solid rocket motors as the propulsion of sounding rockets and satellite launchers. Major projects are the VLS and VLM launch vehicles described e. g. in Isakowitz et al (1999). The solid rocket motor cases for the upper stages of these vehicles are manufactured from composite materials by filament winding to achieve minimum weight and maximum payload. These composite cases are currently analyzed in cooperation with the Department for Mathematical Models in Material Science (MMW) of the Aachen University of Technology (RWTH). The focus of this investigation are the failure modes under internal pressure and the prediction of the burst pressure. These are decisive factors in the design of light-weight pressure vessels. A proper understanding of these factors will lead to optimized designs for future composite rocket motor cases and pressure vessels.

2. PRINCIPLES OF DESIGN - MEMBRANE THEORY AND NETTING ANALYSIS

Historically composite pressure vessels are designed by use of the so called netting analysis. In this approach a membrane shell theory is coupled with the assumption that only the fibers carry the load in the composite material. Pressure vessels for rocket motor cases manufactured by filament winding require some kind of end-closure or dome. This dome cannot be a closed one, but must have openings to be of any practical utilization. These openings are called polar bosses and represent an area where other structures will be attached to the vessel by means of flanges. The smaller opening is designed to receive the igniter that initiates the solid propellant combustion. The remaining opening is designed to be attached to the nozzle, through which the hot gases produced by the burning propellant will expand. The design of the domes is the main part of the design of a composite case. Following the principles of netting theory, the fibers are the main responsible to carry the in-plane loads, whilst the resin contribution is neglected. Since the primary tensile strength and stiffness of the composite derives from the fibers, this theory furnishes an acceptable approximation, especially if one considers that the fiber volume fraction in a filament-wounded composite case ranges from 65 to 70 %.

The design starts with consideration of the membrane forces in meridional and circumferential direction of a shell of revolution submitted to internal pressure p:

$$\frac{n_\theta}{r_\theta} + \frac{n_\phi}{r_\phi} = p \tag{1}$$

$$n_\phi = \frac{1}{\sin\phi} \int_0^\phi p \ \cos\alpha \ d\alpha \tag{2}$$

Equation (1) is derived from the equilibrium of a surface element in the local normal direction. n_θ and n_ϕ are the membrane forces in the meridian and circumferential direction. r_θ and r_ϕ are the related principal radii of curvature of the shell. Equation (2) is obtained from the equilibrium for any part of the dome in axial direction and can be further evaluated to Eq. (3). Substitution of Eq. (3) into (1) yields Eq. (4) so that two independent equations for the membrane forces n_θ and n_ϕ are available.

$$n_\phi = \frac{p\ r_\theta}{2} \tag{3}$$

$$n_\theta = \frac{p\ r_\theta}{2} \left(2 - \frac{r_\theta}{r_\phi}\right) \tag{4}$$

The equilibrium of a balanced lay-up of $+\alpha$ and $-\alpha$ layers can be expressed according to netting theory, where n_f is the membrane force in the local fiber direction with an angle α:

$$n_\theta = n_f \sin^2\alpha \tag{5}$$

$$n_\phi = n_f \cos^2\alpha \tag{6}$$

Combination of Eq. (3) to (6) yields

$$\tan^2\alpha = 2 - \frac{r_\theta}{r_\phi} \tag{7}$$

The principal radii r_θ and r_ϕ can be expressed in carte Sian coordinates once the dome contour is written in terms of the radius r as a function of the axial coordinate z

$$r_\theta = r\left(1 + r'^2\right)^{1/2} \tag{8}$$

$$r_\phi = \frac{(1 + r'^2)^{3/2}}{r''} \tag{9}$$

and a differential equation for the dome contour r depending on the local fiber angle α is obtained:

$$r'' = \left(\tan^2\alpha - 2\right)\frac{1 + r'^2}{r} \tag{10}$$

Rocket motor cases frequently have polar bosses with unequal diameters, because the design specification of nozzle and igniter are different. This condition does not allow the application of an isotensoidal design. A good alternative to the isotensoidal design that allows a stable winding with unequal polar bosses is the planar winding, also called polar winding or "in-plane" winding.

The winding operation of a circuit takes place within a plane defined by the two polar bosses. The following geometrical relationship for the winding angle can be established (Hartung, 1963):

$$\tan^2\alpha = \frac{\left[r\tan\gamma - r'(b + z\tan\gamma)\right]^2}{\left[r^2 - (b + z\tan\gamma)^2\right](1 + r'^2)} \tag{11}$$

In this γ is the inclination of the winding plane with respect to the centerline and b is the distance of this plane with the centerline at the intersection of dome and cylindrical part ($z = 0$). Substituting Eq. (11) into Eq. (10) the following differential equation is derived:

$$r'' = \frac{\left[r\tan\gamma - r'(b + z\tan\gamma)\right]^2}{\left[r^3 - r(b + z\tan\gamma)^2\right]} - \frac{2\,(1 + r'^2)}{r} \tag{12}$$

The numerical solution of Eq. (12) subject to the initial conditions $r(0) = r_0$ and $r'(0) = 0$ determines the dome contour. The local winding angle $\alpha(z)$ can than be evaluated using Eq. (11). In this way a dome may be designed that can carry the internal pressure load by way of membrane forces given the assumptions of the netting theory.

3. FINITE ELEMENT ANALYSIS OF COMPOSITE ROCKET MOTOR CASES

A design found by netting analysis is frequently further analyzed using finite element methods. This is done to address the shortcomings of the underlying membrane shell theory, namely the inability to take bending stresses into account. The finite element analysis are undertaken with different objectives: To verify the design by netting analysis, to give hints for the dimensioning of section thicknesses or to guide local design optimizations. Owing to the different objectives and because of limited computational resources, the complexity and level of accuracy of the finite element models used for the analysis of composite pressure vessels varies greatly. For many years finite element models with axisymmetric or three dimensional layered shell elements or with axisymmetric orthotropic volume elements were the only option for the full scale analysis of composite pressure vessels. Today three dimensional models are affordable and promise a more realistic representation of the structure and consequentially a higher accuracy of the analysis.

3.1. Modeling Options

Several examples for finite element analyses of composite rocket motor cases or composite pressure vessels can be found in the literature. The following authors discuss the bursting problem under internal pressure:

- Gramoll et al (1990): Axisymmetric quadratic volume elements are used for the analysis of a solid rocket motor case. Three elements in thickness direction are used, but it is not clear how the lay-up was represented in the model.
- Levy Neto (1992): Axisymmetric layered shell elements are used to analyze an abstracted pressure vessel with constant fiber angle, constant wall thickness and hemispherical domes. The computational cost of this analysis is quite low.
- Doh and Hong (1995): Three-dimensional quadratic layered shell elements are specifically developed to treat the bursting problem of a composite motor case. A $90°$ sector model is analyzed using symmetry displacement boundary conditions.
- Pereira and Palmerio (1998): Three-dimensional quadratic layered shell elements are used for the analysis of a composite case. A $90°$ sector model with symmetry displacement boundary conditions is considered.
- Sun et al (1999): Axisymmetric volume elements are specifically developed for the analysis of a composite rocket motor case. The number of elements in thickness direction and the representation of the lay-up are not discussed.

This account makes no claim of completeness, but it can be seen that mostly either axisymmetric elements or layered shell elements were used in the past. All references consider an axisymmetric structure and state that a geometrically nonlinear analysis is necessary because of the relatively large displacement encountered for thin walled pressure vessels.

Axisymmetric volume elements which accept orthotropic material properties are widely available and each composite layer could be represented by one or more of these elements. However, due to the large number of layers frequently used in laminated structures, this could lead to an excessive number of elements. Therefore most authors use equivalent orthotropic material properties for elements representing multiple layers (Shu, 1995). A drawback of axisymmetric elements is that only axisymmetric load cases and failure modes can be represented. This is sufficient for the analysis of the bursting problem under internal pressure, but prevents the analysis of other important load cases like overall bending.

Layered shell elements are attractive because very efficient models for thin walled structures are possible. One of the first examples for a layered shell element is given by Panda and Natarajan (1976). The use of layered shell elements for the analysis of motor cases is difficult because critical areas like skirt attachments and the polar regions with strongly varying thicknesses can not be represented easily (Gramoll, 1990; Pereira et al, 1998).

An alternative to the approaches discussed so far is the use of three-dimensional volume elements. Elements with orthotropic properties could be used to resolve individual layers or to represent multiple layers using equivalent properties. This would lead either to an excessive number of elements or to a cumbersome pre- and post-processing. A good compromise is the use of layered volume elements. One of the first of these elements is described by Hoa et al (1985). Three-dimensional volume elements offer the greatest modeling flexibility but lead obviously to the highest computational effort. This fact prevented their use for the full-scale analysis of composite pressure vessels in the past.

3.2. High-Resolution Finite Element Model

The element used in this study is the Solid 46 element available in the well known ANSYS program. This layered volume element is a linear eight node hexaeder with additional incompatible quadratic shape functions to improve the bending behavior. For the through-thickness integration a formulation similar to the one used for the layered shell element discussed in Yunus et al (1989) is employed. This element type was selected because the objective of this study is the detailed simulation of failure modes and the accurate prediction of the burst pressure. Therefore regions of geometric complexity must be represented adequately.

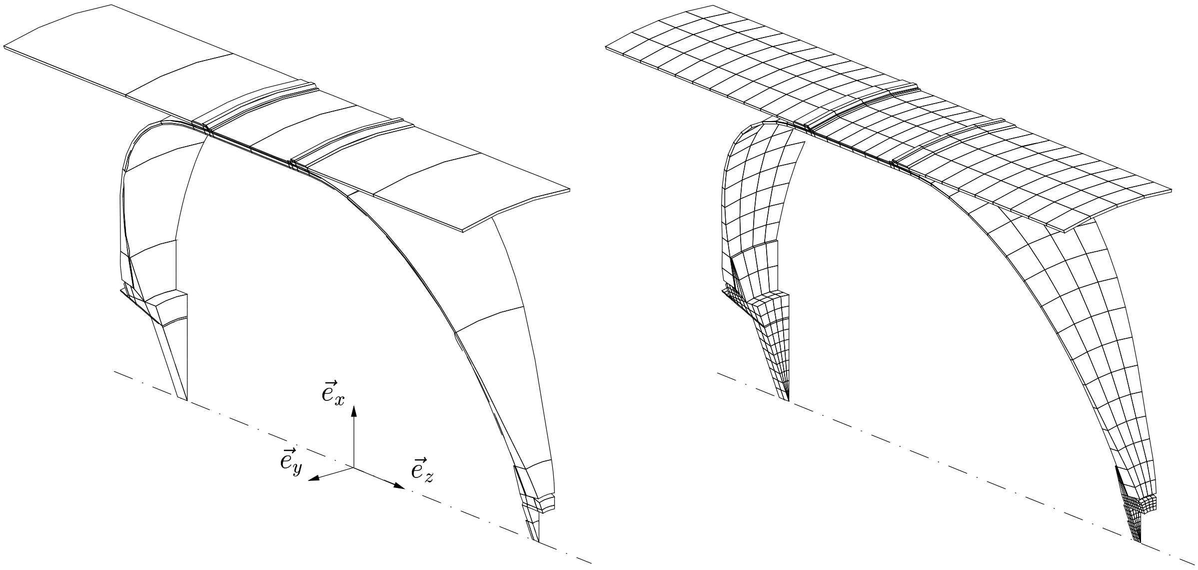

Figure 1. Solid geometry model and finite element mesh ($20°$ sector)

An important feature of the modeling concept underlying the present study is that the finite element mesh is flexibly created by an automated preprocessor on the basis of user-specified parameters. The parameters are defined by the geometry of the pressure vessel to be analyzed and the mesh density of the desired finite element model. The last aspect is essential because it allows the user to scale the computational effort with respect to the required accuracy as described later. The models for the present study are created in three basic steps: In the first step, contour, winding angle and thickness distributions of the pressure vessel are calculated on the basis of netting analysis equations for either planar or geodetic winding and using theoretical or empirical models for the wall thickness distribution. These calculations depend on user-specified parameters like diameter, length and filament band width. The results of this step are collected in data tables. In the second step, a Solid Geometry model of the pressure vessel is created on the basis of the data tables from the first step and additional parameters selected by the user. This geometry model consists mainly of volume entities that can be easily mapped on the unit cube. In the third step, the finite element model consisting of structured meshes is created by mapped meshing. Structured meshes are most appropriate for the selected layered volume elements. Mesh densities can be controlled by the user through the selection of parameters. While mesh densities in meridian and circumferential directions can be selected from a nearly arbitrary range, this is currently not true in the thickness direction: The mesh density in the thickness direction depends on the lay-up because it is undesirable to split layers between elements. Nodes should be placed on layer interfaces instead. For the lay-up of the motor case studied in the present work, either one or five elements in thickness direction can be selected. Figure (1) shows a solid geometry model (left) and the associated finite element model (right). In this case, a $20°$ sector model with four elements in circumferential and one element in thickness direction was created by the preprocessor. More information concerning further features of the model are presented by Krieger et al (2001).

3.3. Mesh Refinement and Convergence

For every finite element analysis the question of the required mesh density with respect to the desired accuracy of the analysis and the corresponding computational cost is raised. To address this question for the present problem a comprehensive convergence study was conducted (Multhoff et al, 2001). Since the preprocessor already described can create different meshes for the same geometry this study was possible with very limited effort. The designation and parameters for the different meshes are listed in Tab. (1).

Table 1. Investigated mesh variants.

	M1	M2	M3	M4	M5	M6	M7	M8
Elements in thickness direction	1	1	1	1	5	5	5	5
Angle/Element in Dome (°)	5	2	0.91	0.45	5	2	0.91	0.45
Number of DOFs	1227	1905	3237	5667	1767	3105	5721	10527
Number of Nodes	1155	1607	2495	4115	1515	2407	4151	7355
Number of Elements	493	634	913	1421	669	1030	1737	3037
Analysis Time (Relative)	1	1.3	2.1	3.8	2.9	3.2	4.9	9.0

Two series of meshes were investigated: The composite dome parts were discretized with either one or five elements in thickness direction. In each step of the mesh refinement the angle spanned by one element in the meridian direction was reduced by approximately 50 %. Meshes with one element spanning an angle of $\theta = 0.5°$ in circumferential direction were used. This selection is based on a separate convergence study varying this angle for the meshes M1 and M8. For the coarse mesh M1, models with $\theta = 0.5°$ and $\theta = 2.0°$ give the same results. A mesh with $\theta = 10°$ gives about the same results for the displacement components u_r and u_z but u_θ is increased by a factor of up to 10. The stress component in fiber direction σ_{11} will however change by less than 2 %. For the fine mesh M8, similar relations hold when meshes with $\theta = 0.5°$ and $\theta = 2.0°$ are compared.

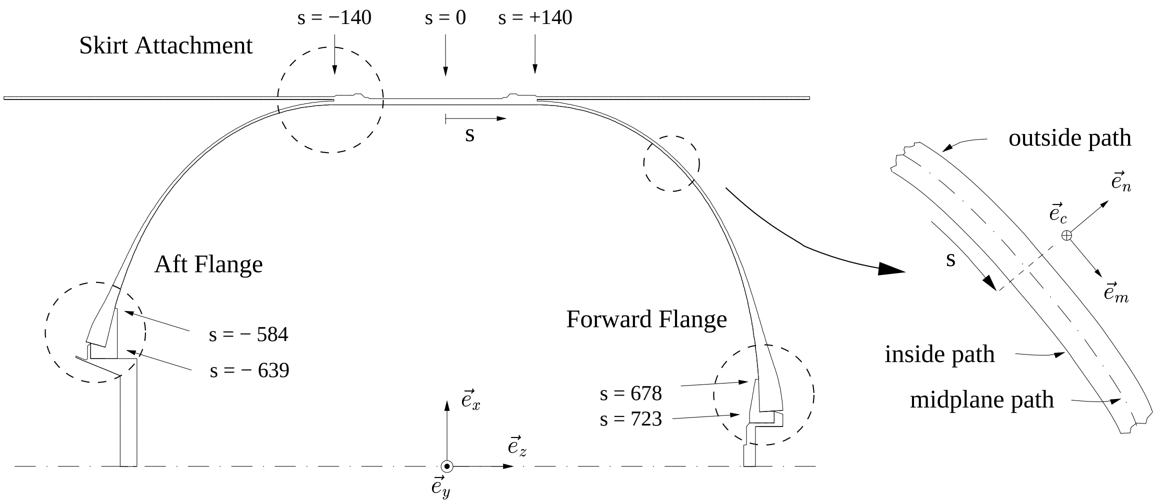

Figure 2. Meridian cut of the motor case

A meridian cut of the motor case is shown in Fig. (2). The critical regions of aft flange, forward flange and skirt attachments are highlighted. Coordinates along the path on the inner surface (winding surface) are marked. Note the coordinates for flange tips and skirt attachments as well as the definition for inside, midplane and outside paths when referring to the result plots presented in the sequel.

Figure (3) compares selected analysis results obtained with the coarse mesh M1 (green marks) and the fine mesh M8 (red lines). All results are displacement or stress components along the inside path of the composite shell as indicated in Fig. (2) for an internal pressure of 11 MPa. The displacement components u_n and u_c are taken with respect to the local coordinate system $(\vec{e}_m, \vec{e}_c, \vec{e}_n)$ indicated in Fig. (2) and are displacements in the direction normal to the shell and the circumferential directions, respectively. A coordinate system aligned with the fibers is used to take stress components. For this purpose the local $(\vec{e}_m, \vec{e}_c, \vec{e}_n)$ system is rotated around the $\vec{e}_n$ vector by the local fiber angle. The stress components σ_{11}, σ_{22} and σ_{12} are the stresses in fiber direction and the in-plane transverse and shear stresses, respectively. The components σ_{33}, σ_{13} and σ_{23} are the out-of-plane transverse and shear stresses.

It is remarkable how well the results for the coarse and fine meshes agree in general. However, the out-of-plane stress components are not predicted correctly by mesh M1. This is due to the fact that

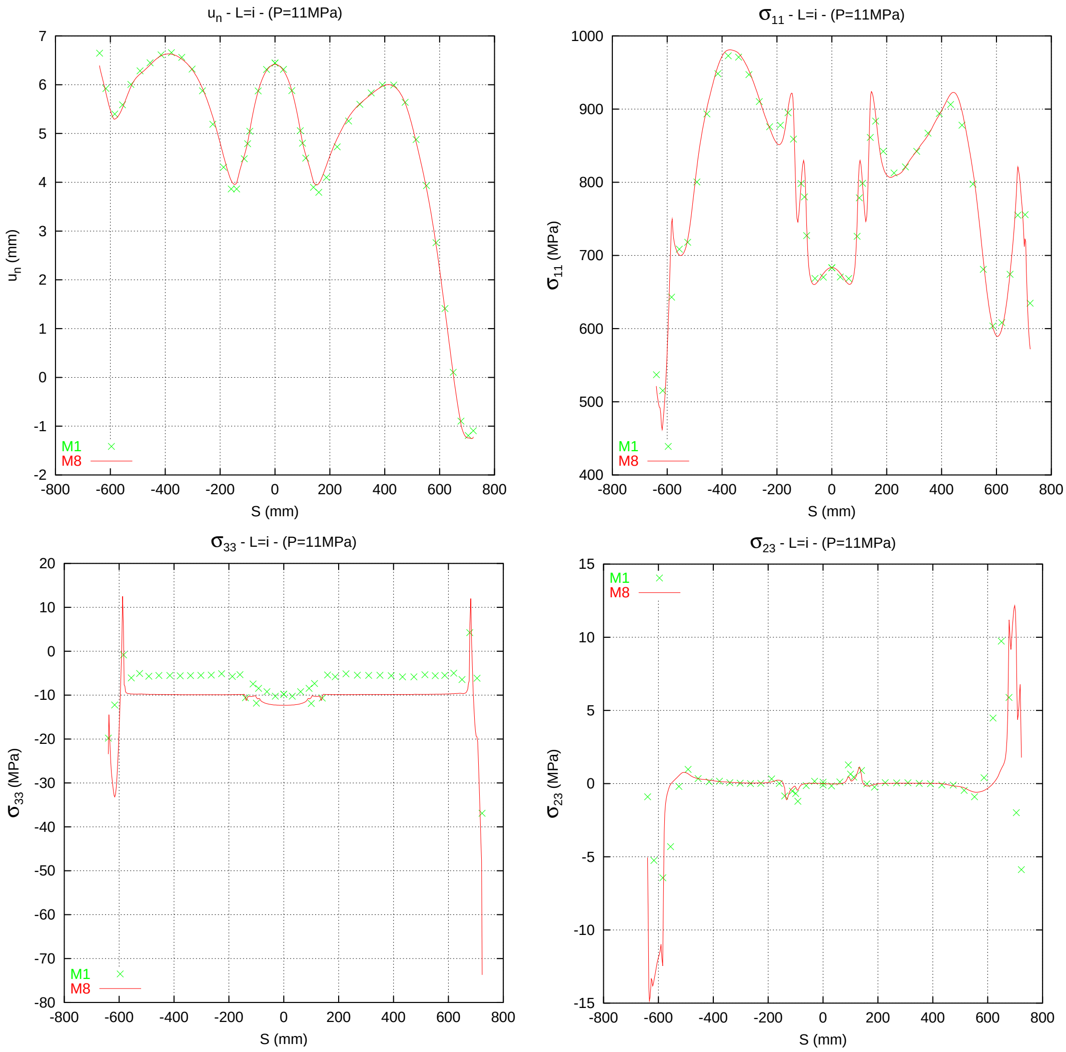

Figure 3. Results for coarse mesh M1 and fine mesh M8

only one element in thickness direction is used. Compare the shear stress σ_{23} in the flange region and note the deviation of the normal stress component σ_{33}. In the dome regions the boundary condition $\sigma_{33} = -p = -11$ MPa should be recovered. It must also be noted that mesh M1 is too coarse to represent all features of the stress distributions present in mesh M8.

4. EXAMPLE ANALYSIS

The finite element model was applied to analyze the behavior of the S44 rocket motor case used in the Brazilian space program (Krieger et al, 2001). This was done by simulating the hydrostatic test used to verify such motor cases (Loures et al, 2001).

As the S44 rocket motor case is manufactured in wet filament winding, the resin must be cured at an elevated temperature to form a solid structure. The cure process is followed by a cool-down to room temperature leading to a deformed contour. The first step in the finite element analysis is therefore to simulate the cool-down from the curing temperature of $120°C$ to room temperature of $25°C$ by application of a temperature load of $-95°C$. The geometrically nonlinear behavior is taken into account and it is assumed that at the end of the curing process the vessel can deform freely and no residual stresses are present. Following the actual production sequence, the aluminum skirts and closures are not installed in this part of the simulation.

The deformed structure after cool-down is taken as the initial geometry for the subsequent pressure loading in the second part of the simulation. The aluminum closures and skirts are now incorporated. The pressure load is applied in steps of 0.5 MPa and the computation again takes the geometrically nonlinear

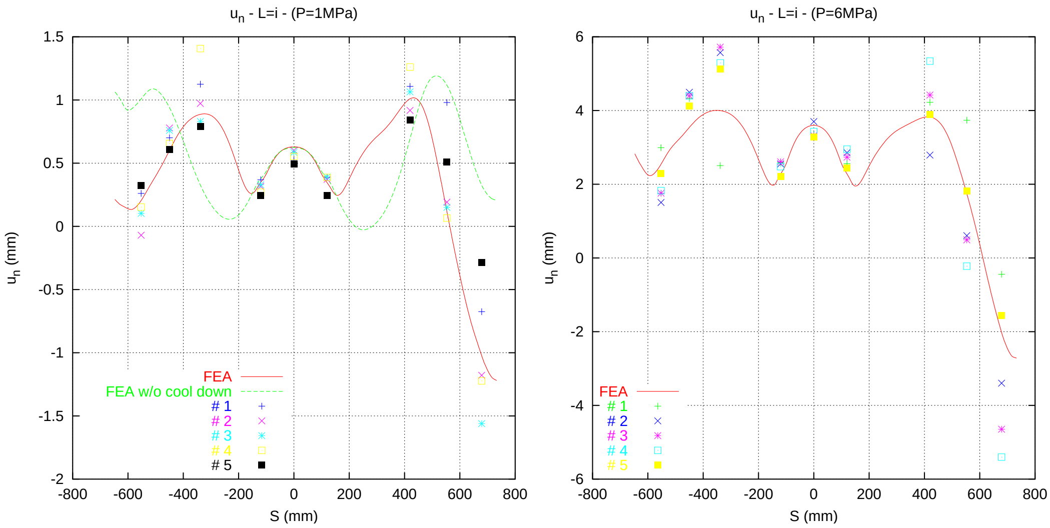

Figure 4. Measured and predicted displacements for 1 MPa (left) and 6 MPa (right) internal pressure

behavior into account. The entire simulation uses for the dome areas the same high mesh density with five elements in the thickness direction and an average element length along the meridian of $0.5°$. This corresponds to mesh M8 used in the convergence study.

The computed displacements for the different load steps can be compared with measured deformations. These measurements were taken during hydraulic tests of different vessel specimens using displacement transducers. In each case the transducers were located at specific points on the vessel and oriented perpendicular to the local surface, so that displacements in the normal direction are measured (Loures et al, 2001). The measured displacements are in a very good agreement with the simulation for 1 MPa internal pressure as can be seen in Fig. (4) on the left. Notably an inward movement of the forward dome is both predicted (red line) and actually measured (marks for 5 different specimen). However, the computation predicts a totally different deformation behavior if the cool-down simulation is skipped (green line). Measured and predicted displacements for 6 MPa internal pressure are compared in Fig. (4) on the right side. Again, the agreement is acceptable, especially if one takes the higher variability of the measured results into account. Comparison of the computed curves of 1 MPa and 6 MPa internal pressure confirms the expected geometrically nonlinear character of the structural deformation.

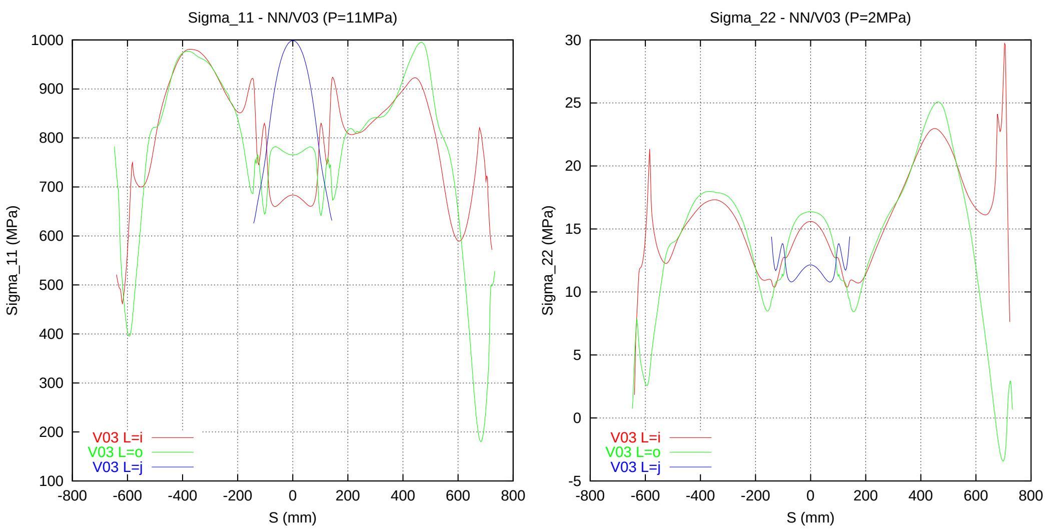

Figure 5. Computed stresses in fiber direction (left) and in transverse direction (right)

The distribution along the meridian of the stress component in fiber direction for 11 MPa internal pressure is shown in Fig. (5) on the left for three different paths in the composite material. Path ’i’ represents the inside of the innermost helical composite layer extending from opening to opening with a varying fiber angle α. Path 'o' belongs to the outside of the corresponding outermost helical layer. Path 'j' is on the inside of the innermost hoop layer which is restricted to the cylindrical part having a fiber angle of $90°$. In the middle of the vessel bending exists which can be determined from the different stress levels in the inside 'i' and the outside 'o' paths in the helical layers. The cylinder length is too short to allow the development of a bending free state away from the disturbance due to the dome and skirt attachments. Within the aft dome a nearly perfect membrane stress state is reached with a maximum stress exceeding the bending dominated stress in the region of the dome/cylinder junction. At the location of the aft flange tip the stress on the inside path 'i' shows a singularity due to the reentrant corner as shown in Fig. (2). In the middle of the forward dome a region of local bending disturbs the membrane stress state. The hoop layers in the cylindrical part, represented by the path 'j', have the maximum stress in the middle of the vessel, which is comparable to the maximum stresses within the dome regions. The very similar stress maxima for the helical and hoop layers demonstrate that the guiding rule for the vessel design is fulfilled: The stresses in fiber direction are balanced.

However, the assessment of the stress state has to include possible differences of the material strength in the different regions of the motor case. Even though the same material constituents are used, the final composite material is created through the winding process and the subsequent curing stage. Due to the missing hoop windings within the dome areas, the compaction of the helical layers is lower and the fiber volume fraction might be decreased by up to 10 % (Peters, 1987). The winding pattern of the $\pm\alpha$ helical-wound layers with the resulting fiber crossing/weaving effects also reduces the strength relative to the hoop-wound layers (Gramoll, 1990). These effects result in a reduced strength within the dome areas. Thus the higher stresses within the hoop layer is not relevant for the failure. The initiation of failure must be assumed to happen in the dome areas. In fact the failure of the different vessel specimen seam all to have occurred in the middle of the aft dome region. The average burst pressure of 10.6 MPa for these specimen is comparable to the presented evaluation of the stress state at 11 MPa.

Another notable result are the stresses transverse to the fiber direction as shown in Fig. (5) on the right for an internal pressure of 2 MPa. This suggests that matrix cracking starts at this pressure level in the middle of the forward dome and subsequently spreads into other areas. This matrix cracking leads to reduced material stiffness and strength which is not taken into account in the present analysis.

5. CONCLUSIONS

Full-scale high-resolution models of composite solid rocket motor cases can be build using existing layered volume elements. Axisymmetric coupling conditions are useful to enforce an axisymmetric solution without artificial stress perturbations. One element in circumferential direction is sufficient, which reduces the computational effort and makes these models affordable for use on personal computers even for nonlinear analyses. If only in-plane stress components are of interest, the use of one element in thickness direction is sufficient to represent the composite dome areas. For the resolution of out-of-plane stresses, especially in the flange areas, multiple elements in thickness directions are necessary. A finite element model for composite pressure vessels is developed to enable a detailed stress analysis under the load of internal pressure. This model is parameterized and can therefore be applied to different types of pressure vessels and also for sensitivity studies. The model is used to analyze a specific rocket motor case and verified by comparison with deformation measurements obtained in hydrostatic tests of specimen. The deformation of the specimen vessels is matched, if the geometry changes from the cool-down after curing are taken into account. The analysis gives detailed information about the mechanical behavior that improves understanding of the failure process and can be used for optimizations.

6. ACKNOWLEDGMENT

This work was supported by a research grant of the German National Science Foundation (DFG) under project number Be 766/40-1. The bilateral cooperation between IAE and MMW is supported by the International Bureau of the German Ministry for Education and Research (BMBF) at the German Aerospace Center (DLR) under project number BRA 99/02. This support is gratefully acknowledged.

7. REFERENCES

Doh, Y.D. and Hong, C.S., 1995, ”Progressive Failure Analysis for Filament Wound Pressure Vessel”, J. of Reinforced Plastics and Composites, Vol. 14, pp. 1278-1306

Gramoll, K.C., Namiki, F. and Onoda, J., 1990, ”General Structural Analysis of an Upper Stage Composite Rocket Motor Case”, Proceedings of the 17th International Symposium of Space Technology and Science, Tokyo, Japan

Hartung, R.F., 1963, ”Planar-Wound Filament Pressure Vessels”, AIAA Journal, Vol. 1, Dec 1963, pp. 2842-2844

Hoa, S.V., Yu, C.W. and Sankar, T.S., 1985, ”Analysis of Filament Wound Vessel Using Finite Elements”, Composite Structures, Vol. 3, pp. 1-18

Isakowitz, S.J., Hopkins, J.P. and Hopkins J.B., 1999, ”International Reference Guide To Space Launch Systems”, AIAA, pp. 487-500

Jones, R.M., 1999, ”Mechanics of Composite Materials”, 2nd ed., Taylor & Francis, Philadelphia

Krieger, J., Multhoff, J.B., Loures da Costa, L.E.V. and Betten, J., 2001, ”Development of a Finite Element Model for Composite Pressure Vessels and its Application in Structural Optimization”, Proceedings of COBEM 2001, Aerospace Engineering, Vol. 6, pp. 366-375

Levy Neto, F., 1992, ”Finite Element Simulation of Composite Pressure Vessels”, Proceedings of the 7th Brazilian Symposium on Piping and Pressure Vessels, Florianópolis, Brazil

Loures da Costa, L.E.V., Multhoff, J.B., Krieger, J. and Betten, J., 2001, ”Structural Development of the S33 Rocket Motor Case”, Proceedings of COBEM 2001, Aerospace Engineering, Vol. 6, pp. 237-246

Multhoff, J.B., Krieger, J., Loures da Costa, L.E.V. and Betten, J., 2001, ”Problems in High-Resolution Finite Element Models of Composite Rocket Motor Cases”, Proceedings of COBEM 2001, Aerospace Engineering, Vol. 6, pp. 207-216

Panda, S.C. and Natarajan, R., 1976, ”Finite Element Analysis of Laminated Shells of Revolution”, Computers & Structures, Vol. 6, 61

Pereira, J.C. and Palmerio, A., 1998, ”Failure Analysis of a Filament-Wound Pressure Vessel”, Proceedings of the ANSYS Conference, Pittsburgh, USA

Peters, S.T. and Humphrey, W.D., 1987, ”Filament Winding”, Engineering Materials Handbook, Composites, Ed. T.J. Reinhard, 3rd ed., ASM International

Rosato, D.V. and Grove, C.S., 1964, ”Filament Winding: Its development, manufacture, application, and design”, John Wiley & Sons, New York

Shu, J.C., Chiu, S.T. and Chang, J.B., 1995, ”An Enhanced Analysis Method for Composite Overwrapped Pressure Vessels”, AIAA, Proceedings of the 36th Structures, Structural Dynamics and Materials Conference, New Orleans, pp. 394-403

Sun, X.-K., Du, S.-Y. and Wang, G.-D., 1999, ”Bursting problem of filament wound composite pressure vessels”, Int. J. of Pressure Vessels and Piping, Vol. 76, pp. 55-59

Yunus, S.M., Kohnke, P.C. and Saigal, S., 1989, ”An Efficient Through-Thickness Integration Scheme in an Unlimited Layer Doubly Curved Isoparametric Composite Shell Element”, Int. J. for Num. Meth. in Engineering, 28, pp. 2777-2793

8. COPYRIGHT NOTICE